Canada - Ontario Mineral Development Agreement

Entente Canada - Ontario sur l'exploration minérale

Canada

Ontario

CANADA - ONTARIO
MINERAL DEVELOPMENT AGREEMENT

SIMULATION OF BULK MINING AT DEPTH WITH BACKFILL IN ONTARIO MINES

Final Report — Volume 6

MINE3D USER'S MANUAL

by

Mines Research Department
INCO Limited

June 1991

91 pp

Work on the project was conducted under the auspices of the Canada Centre for Mineral and Energy Technology, Energy, Mines and Resources Canada under the Canada/Ontario Mineral Development Agreement.

DSS File No. 14SQ.23440-6-9033
DSS Contract Serial No. 23440-6-9033/01-SQ
CANMET Project No. 6-9033
CANMET Scientific Authority: Y. S. Yu

MICROMEDIA

93-01530/

ABSTRACT

MINE3D is a three-dimensional boundary element program which calculates stresses, displacements and failure zones resulting from the simulation of bulk mining in massive orebodies. The program includes both a plasticity formulation to simulate localized failure regions around mined openings and a multiple material formulation to model regions of different materials such as backfill within the orebody.

MINE3D consists of three separate computer codes: a graphical preprocessor MIP (MINE3D Input Processor), a main calculational program MAP (MINE3D Analysis Program), and a graphical postprocessor MOP (MINE3D Output Processor). This volume is the User's Manual for MINE3D and explains the operation of MIP, MAP and MOP. Example analyses are also presented to assist the user.

RÉSUMÉ

MINE3D est un programme d' éléments qui calcule les contraintes, les déplacements, et zones de rupture résultant de la simulation du minage en vrac des gisements massifs. Le programme inclut une formulation en plasticité pour simuler la présence de régions de rupture locale autour des excavations miniéres, et permet l'introduction de matériaux multiples pour représenter les régions de matériaux aux différents tels le remblai à l'intérieur du gisement.

MINE3D consiste en trois (3) codes informatiques séparés: le pré-processeur graphique MIP (le module des données de MINE3D), le module principal de calcul MAP (le programme d'analyse de MINE3D), et le post processeur MOP (le module des résultats de MINE3D). Ce rapport est le Manuel de lúsager du logiciel MINE3D et explique le fonctionnement des modules MIP, MAP et MOP. Des exemples dánalyses sont également présentés pour assister l'usager.

PREFACE

This project, entitled "Simulation of Bulk Mining at Depth in Ontario Mines with Backfill," consists of two major components: the development of a numerical model, and field verification of the model. The project, which began in 1986, was expected to take four years, ending in September 1990, but was extended until March 1991. This report is the sixth of six volumes which describes the results of the project.

The work was initiated by INCO Limited and is partially funded through the MDA Program (joint Canada-Ontario Mineral Development Agreement, DSS file 14SQ.23440-6-9033). The project is administered by CANMET and its assistance is greatly appreciated.

LIMITATION OF LIABILITY

CANMET, INCO Limited, and their subcontractors assume no liability whatsoever with respect to any use of MINE3D or any portion thereof, or with respect to any damages or losses which may result from such use, including (without limitation) loss of time, money or goodwill which may arise from the use of MINE3D. In no event shall CANMET, INCO Limited, or their subcontractors be responsible for any indirect, special, incidental or consequential damages arising from the use of MINE3D.

TABLE OF CONTENTS

PAGE

ABSTRACT i

RÉSUMÉ ii

PREFACE iii

LIST OF FIGURES ix

LIST OF TABLES xii

1.0 INTRODUCTION 1

2.0 THE BOUNDARY ELEMENT METHOD IN MINE3D 2

2.1 Boundary Element Formulation 2

2.2 Modeling Plasticity 3

2.3 Inclusion of Multiple Regions 4

2.4 Sequential Mining 5

2.5 Solution Process 6

2.6 Boundary Element Geometry 7

3.0 NUMERICAL MODELING WITH MINE3D 11

3.1 Software Overview 11

3.2 Program Requirements and Limitations 11

3.3 Modeling Sequence 13

3.4 Save/Restart Procedure 14

3.5 Energy Calculation 15

TABLE OF CONTENTS
(continued)

PAGE

3.6 Sign Convention and Units *16*

3.7 Hard-Copy Plotting *16*

4.0 MINE3D INPUT PROCESSOR (MIP) **18**

4.1 Initial Entry to MIP *18*

4.2 MIP's Main Menu *19*

4.2.1 Mining Steps *20*

4.2.2 Print Request *20*

4.2.3 Reading and Writing Files *20*

4.2.4 Properties, Parameter and Options *21*

4.2.5 AutoCAD Layers *21*

5.0 MINE3D ANALYSIS PROGRAM (MAP) **23**

5.1 Element and Node Definitions *23*

5.2 Input Data File *27*

5.2.1 Problem Heading *27*

5.2.2 Solution Parameters *27*

5.2.3 Parallelepiped Nodes *28*

5.2.4 Global Coordinates for Parallelepiped Grid *28*

5.2.5 Far-Field Stress Parameters for Each Region *28*

5.2.6 Material Properties for Each Material *28*

5.2.7 Surface Element List *29*

5.2.8 Surface Node List *29*

5.2.9 Volume Element List *29*

5.2.10 Volume Node List *29*

TABLE OF CONTENTS
(continued)

PAGE

5.2.11 List of Surface Elements Turned On or Off for This Mining Step 29
5.2.12 List of Surface Nodes Turned On or Off for This Mining Step ... 29
5.2.13 List of Volume Elements Turned On or Off for This Mining Step 30
5.2.14 List of Volume Nodes Turned On or Off for This Mining Step ... 30
5.2.15 List of Connected Nodes for This Mining Step 30
5.2.16 List of Altered Nodes for This Mining Step 30
5.2.17 Volume of Region Excavated for This Mining Step 30
5.2.18 Selected Output Plane 30

6.0 MINE3D OUTPUT PROCESSOR (MOP) **36**
6.1 Menu Window ***36***
6.2 Graphics Window ***38***
6.2.1 Model Positioning *38*
6.2.2 Multiple Materials and Regions *39*
6.2.3 Calculated Data Plots *39*
6.2.4 Contour Limits *39*
6.2.5 Displacements *40*
6.2.6 Function Keys *40*

7.0 SOFTWARE PROGRAM GUIDE **42**

8.0 EXAMPLE PROBLEMS **50**
8.1 Square-Shaped Drift ***50***
8.2 Excavation / Fill Stopes ***66***

TABLE OF CONTENTS
(continued)

PAGE

9.0 REFERENCES **80**

APPENDIX I: MINE3D SOURCE FILES **I-1**

APPENDIX II: COLOR PLOTS FROM PostScript(R) FILES **II-1**

LIST OF FIGURES

PAGE

Fig. 1 An Inclusion As Two Separate Problems 5

Fig. 2 A Parallelepiped Grid Overlaid On a Stoping Layout 8

Fig. 3 Coefficient Matrix Size versus Number of Boundary Element Nodes 9

Fig. 4 Comparison of MINE3D Iteration Rate versus Number of Surface Nodes for Different Computer Systems 10

Fig. 5 Example 3-D Perspective Plot from MOP for a Model Containing 9261 Parallelepiped Nodes and 1748 Surface Nodes 12

Fig. 6 Default Sign Convention for Input and Output Stresses in MINE3D (Note that the outward normal for the x-, y-, and z-planes points in the negative x-, y-, and z-directions.) 17

Fig. 7 Menu Tree for MIP 22

Fig. 8 Surface Node Definition 24

Fig. 9 Surface Element Definition 24

Fig. 10 Volume Element / Node Definition 25

Fig. 11 Integer Code for Nodal Coefficients 25

Fig. 12 Node Codes for a Square Void 26

Fig. 13 MIP Initial Screen Display 53

Fig. 14 Grid Dimensions Menu 54

Fig. 15 Excavation in Slice 1 of x′-y′ Plane of Grid 55

LIST OF FIGURES
(continued)

PAGE

Fig. 16 Excavation in Slice 4 of y′-z′ Plane of Grid **56**

Fig. 17 3-D Geometry Plot from MOP **57**

Fig. 18 Displacement Vectors on K-5 Plane for Elastic Analysis **58**

Fig. 19 σ_3 Stress Contours on K-5 Plane for Elastic Analysis **59**

Fig. 20 Select Element Type Menu **60**

Fig. 21 Excavation and Plastic Elements in Slice 1 of x′-y′ Plane of Grid **61**

Fig. 22 Excavation and Plastic Elements in Slice 4 of y′-z′ Plane of Grid **62**

Fig. 23 Displacement Vectors on K-5 Plane for Plastic Analysis **63**

Fig. 24 σ_3 Stress Contours on K-5 Plane for Plastic Analysis **64**

Fig. 25 Plastic Nodes on K-5 Plane for Plastic Analysis **65**

Fig. 26 Layers Within an AutoCAD "DXF" File **67**

Fig. 27(a) MINE3D Grid Overlays AutoCAD Drawing: View of Full Drawing **68**

Fig. 27(b) MINE3D Grid Overlays AutoCAD Drawing: Close-Up (Zoom-In) View **69**

Fig. 28 Mining Step 1 ..... **70**

Fig. 29 Mining Step 2 ..... **71**

Fig. 30 3-D Geometry Plot of Excavation and Fill Stopes: (a) Mining Step 1; (b) Mining Step 2 ..... **73**

LIST OF FIGURES
(continued)

PAGE

Fig. 31 Displacement Vectors on K-4 Plane for Mining Step 1 ..... **74**

Fig. 32 Displacement Vectors on J-13 Plane for Mining Step 1 ..... **75**

Fig. 33 σ_3 Stress Contours on K-4 Plane For Mining Step 1 ..... **76**

Fig. 34 Displacement Vectors on K-4 Plane For Mining Step 2 ..... **77**

Fig. 35 Displacement Vectors on J-13 Plane For Mining Step 2 ..... **78**

Fig. 36 σ_3 Stress Contours on K-4 Plane For Mining Step 2 ..... **79**

LIST OF TABLES

PAGE

Table 1 SYSTEMS OF UNITS **17**

Table 2 MAP INPUT DATA FILE **31**

Table 3 GRAPHICS WINDOW KEY COMMAND SUMMARY **41**

1.0 INTRODUCTION

MINE3D is a three-dimensional boundary element computer program for calculating stress redistribution around underground mines. This code is designed to simulate large-scale bulk mining with backfill, and contains the following code features to support this capability.

- The code has both linearly-elastic surface elements and plastic volume elements to simulate localized regions of plastic yield within an elastic continuum. Linear strength elements are used to provide greater accuracy for the non-linear analysis.
- Multiple regions can be defined in order to represent different materials, such as backfill within an orebody.
- 3-D element geometries are constructed within a "grid" of parallelepipeds. This permits the analysis of larger problem sizes than can be generated with arbitrary-shaped boundary elements.
- The code uses a graphical preprocessor to generate MINE3D data files. The preprocessor accesses AutoCAD mine layouts which are overlaid on "levels" of the parallelepiped grid. A cursor is used to select elements to be excavated, backfilled, or defined as plastic.

The mathematical formulation and features of MINE3D are described in detail in "Simulation of Bulk Mining at Depth with Backfill in Ontario Mines, Volume 2 — Mathematical Formulations and Code Development." Section 2, below, summarizes these features.

MINE3D operates as three separate programs: a graphical preprocessor, MIP (MINE3D Input Processor), main calculational program, MAP (MINE3D Analysis Program), and a graphical postprocessor, MOP (MINE3D Output Processor). MIP generates an input file for both executing in MAP and for displaying a three-dimensional perspective view of the mining model with MOP. MAP provides an output file to MOP for displaying results from the analysis on sections through the 3-D perspective view.

Both MIP and MOP are menu-driven and greatly simplify model generation and interpretation of results. The operation of these programs and the structure of the input file parameters generated for the MAP calculation are discussed in Sections 3 through 6. A guide to the software data structure, is given in Section 7. Example applications of MINE3D are provided in Section 8.0.

2.0 THE BOUNDARY ELEMENT METHOD IN MINE3D

2.1 *Boundary Element Formulation*

MINE3D is based upon the ***indirect*** boundary element method. The theory behind this method is described in detail by Banerjee and Butterfield (1981). A brief explanation of the concepts is given here so that an appreciation of the application of the approach in MINE3D can be obtained. The formulation is described in detail in Volume 2.

The indirect method uses "fictitious stress" or "fictitious force" elements. Imagine a force applied at an arbitrary point in an initially stress-free medium. This force will cause stresses and displacements to occur at all point in the body. In a linearly-elastic body, the expression for the stresses induced by this force is:

$$\underline{\sigma}(x) = \underline{T}(x,\xi)\ \phi(\xi) \tag{1}$$

That is, the stress (σ) at any point x is related to the forces (ϕ) at a point ξ via the function T, known as the Green function. Similarly, as an area A experiences the body force, then

$$\underline{\sigma}(x) = \int \underline{T}(x,\xi)\ \phi(\xi)\ dA \tag{2}$$

If the area is broken into N sub-areas (or elements) A_n, Eq. (2) becomes

$$\underline{\sigma}(x) = \sum_{n=1}^{N} \int \underline{T}(x,\xi)\ \phi(\xi)\ d\ A_n \tag{3}$$

For "constant strength" elements, the values of ϕ are constant over each element, and Eq. (3) becomes

$$\begin{aligned} \underline{\sigma}(x) &= \sum_{n=1}^{N} \underline{\phi}(\xi) \int \underline{T}(x,\xi)\ d\ A_n \\ &= \left[\mathbf{T}\right] \phi \end{aligned} \tag{4}$$

The magnitudes of the quantities ϕ_n can be related to the tractions t at a specified number of points M:

$$\underline{t}^m = \sum_{n=1}^{N} \underline{B}_n \, \underline{\phi}_n \qquad m = 1,2 \ldots M \tag{5}$$

where B_n can be related to T_n.

If $M = N$, then

$$\underline{t}^m = \sum_{n=1}^{N} \underline{B}_n \, \underline{\phi}_n \qquad m = 1,2 \ldots N \tag{6}$$

or

$$t = [B] \, \phi \tag{7}$$

This provides 3N equations relating the 3N unknown quantities ϕ_n to the 3N known tractions.

Equation (6) is the basis of the indirect boundary element method. If the sub-areas A_n are small areas of excavation boundaries, and the points at which tractions are known are the centers of these areas, by solving Eq. (6), the values of ϕ which are required to satisfy the traction boundary conditions are found. Knowing these, the stress anywhere around the excavation can be calculated from Eq. (4). A similar equation can also be derived to calculate displacements.

The boundary element formulation in MINE3D differs from the description above only in that linear-strength elements are used rather than constant-strength elements. This means that nodes are located at corners of elements rather than at element centers. As a result, better accuracy can be obtained than with constant-strength elements. (See Volume 2.)

2.2 *Modeling Plasticity*

When plasticity is introduced into the method, the term boundary element becomes something of a misnomer because it turns out, as described below, that the yielding medium must also be divided into elements. These volume elements, however, are only introduced in regions where yield occurs. Of course, if the yielded area is not known in advance, a large region can be assumed, or a reasonable guess can be made for the size of the plastic zone.

The mathematical theory required for introducing plasticity is fairly complex, and again, the reference is made to Banerjee and Butterfield (1981) and Volume 2. In the simplest terms, the plasticity model can be described as follows.

The stress at any point, assuming no yield occurs, is given by σ^e. The correct stress, accounting for yield is σ^{ep}. Let $\sigma^o = \sigma^e - \sigma^{ep}$. It can be shown that the problem to be solved by boundary

elements is an elastic one, with the stress anywhere given by $(\sigma^e - \sigma^o)$. This results in a traction boundary condition given by $[(\sigma^e - \sigma^o) \bullet n]$ instead of $(\sigma^e \bullet n)$ and with a body force distribution of the form

$$f_i(x) = -\frac{\partial\, \sigma^o_{ij}(x)}{\partial\, x_j} \tag{8}$$

The effect of a point body force is known from the previously-described elastic method, and the effect of a force distributed over a volume is obtained by integrating over the volume rather than over an area. All that remains, therefore, is to describe how the distribution of σ^o is calculated. This is discussed following the description of multiple regions and sequential mining.

2.3 *Inclusion of Multiple Regions*

The point-force solutions used in the boundary element method described above apply only to excavations in infinite isotropic, linearly-elastic media, or finite solid bodies. The method can be generalized to include multiple materials by treating each material (or region) as an independent problem, and letting the problems provide boundary conditions to each other. In this way, for example, backfill can be simulated in an advancing excavation through its interactions with the surfaces of the surrounding rock mass.

Whenever two regions meet, one region (the finite body) has a displacement boundary condition while the other, infinite region, has a traction boundary. At each calculational interaction, the boundary condition is changed based on the adjacent region. For example, if a surface node in region 1 is adjacent to a surface node in region 2, node 1 may move, for instance, due to an excavation. The displacement which develops at node 1 then acts as a boundary conditions for node 2. Tractions will then develop at node 2 to compensate for this displacement. On the next iteration, node 1 will see these tractions as a boundary conditions. Figure 1 demonstrates the interaction.

Nodes between adjacent regions are "connected" to permit this transference of boundary conditions. The connection is generated automatically by the preprocessor MIP. The preprocessor also assigns the boundary conditions such that the softer material (e.g., backfill) experiences the displacements and returns tractions to the stiffer material. This provides better numerical stability during solution.

The only restriction on the multiple material logic is that the Poisson's ratio for all materials must be equal. In this case, all stress coefficients are unchanged from material to material, and displacement coefficients only differ by the ratio of Young's moduli for different materials. This avoids having to store coefficient sets for all materials separately.

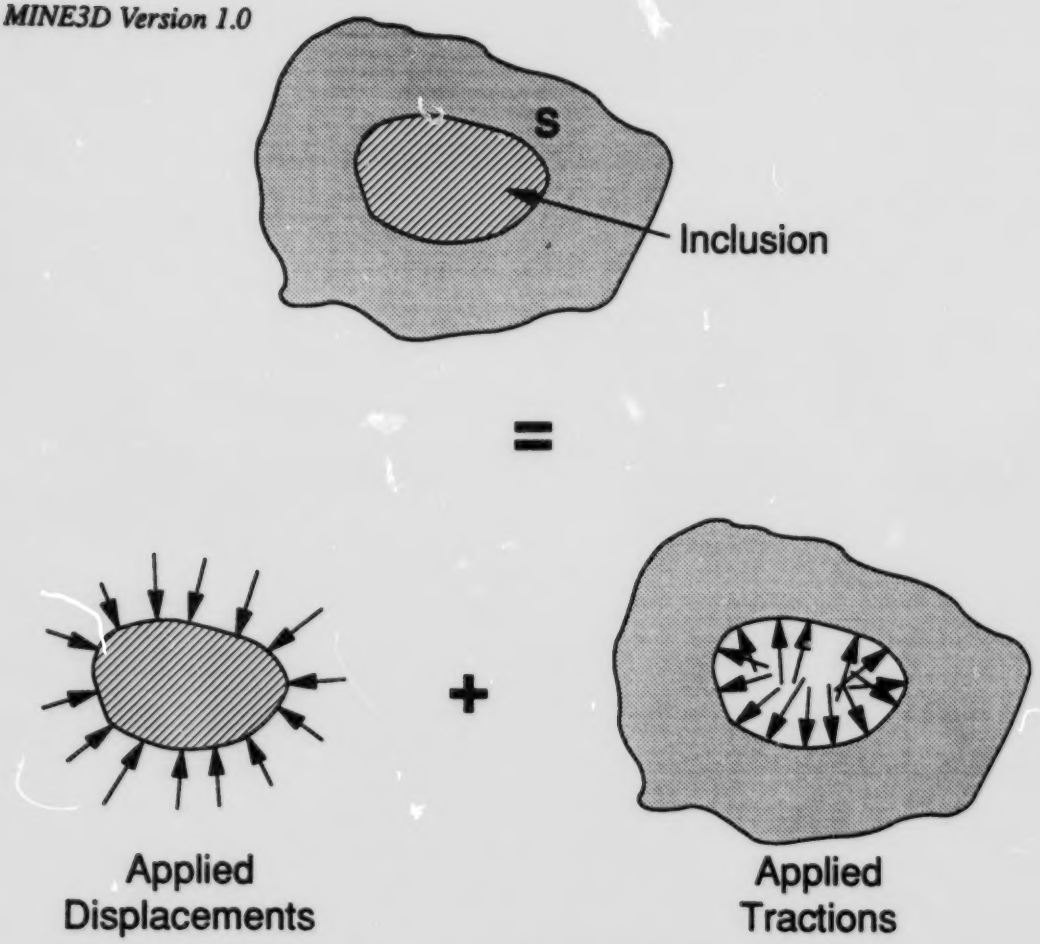


Fig. 1 An Inclusion As Two Separate Problems

2.4 *Sequential Mining*

Sequential mining has been introduced in the code to calculate the stresses and displacements at progressive stages of the mining operation. The input is prepared for a sequence of mining steps. At each new step, existing excavations are enlarged, new ones created, and backfill set in place. Also, regions in which plastic behavior is expected are created or updated. The output solution provides global (rather than incremental) values for stresses and displacements at each step.

Each new step is viewed as a modification of the former one. This eases the input process and speeds up intermediate calculations by providing at the new step "good" first guesses for unknowns involved in the iterative processes based on values obtained at the former step.

When all materials in the medium behave elastically, the solution at a particular mining step may be determined independently. For the case in which plasticity is involved, the mining history is taken into account by using the plastic stress solution from the previous mining step as the initial guess for the plastic initial stress field at the new mining step.

2.5 *Solution Process*

An iterative process is used in the solution of the system of equations, to take into account plasticity and multiple materials. This is described as follows.

For each mining step:

1. Guess the initial plastic stress.

   The value of σ_{ij}° at the nodes of the volume elements defined for a given step is selected based on the value obtained at the previous mining step, if it is available. If not, the value for σ_{ij}° is set to zero.

2. Solve for fictitious forces

   Including the assigned value of σ_{ij}°, three equations are set up for each of the surface nodes which express the fact that three components of traction or displacement are imposed at the node. At a "single" surface node, the value imposed is known beforehand, and corresponds to a boundary condition for the overall problem. At a "twin" node, the value imposed is computed using an iterative procedure devised to ensure continuity of tractions and displacements across the interface. The resulting system of equations is solved, providing values for the fictitious forces.

3. Compute initial plastic stress correction.

   For each volume node, the stresses are computed using both the values for the guess at σ_{ij}° and those for the associated fictitious forces. If the yield criterion is violated at the node, the stresses are brought back to the yield curve by means of a non-associated flow rule. The corresponding plastic stress correction is calculated and stored.

4. Check for convergence.

   If the value of the plastic stress correction falls below a pre-set tolerance at all volume nodes, the solution is obtained. Otherwise, the initial stress field is updated, and the procedure is repeated, beginning at Step 2.

The linear system of equations is solved using the "successive overrelaxation" method (see, for example, Dahlquist and Bjorck, 1974, p.192). This iterative method reduces to the Gauss-Seidel method when the relaxation parameter is selected as one.

2.6 *Boundary Element Geometry*

The theory developed above, and for most currently available boundary element codes, allows elements of arbitrary shape. This means that when the influence coefficients (the components of matrix B in Eq. (7)) are calculated, they are all different. Each elastic surface node N_e has three unique traction components, and their effect is required on three traction components of any other node. Likewise, each plastic volume node N_p has six unique components of σ^o, and their effect is required on six stress components at any other node. Thus, if it is assumed that surface and volume nodes do not coincide, then there are $(3N_e + 6N_p)$ unknowns and an equal number of equations to describe their interaction. There are, then, $(3N_e + 6N_p)^2$ unique coefficients in the matrix if no account is made for common nodes or symmetry.

The storage requirements for these coefficients is enormous. One of the primary purposes for MINE3D is to solve mine-scale size problems. It therefore becomes necessary to reduce storage demands for solution on the workstation or PC environment. In order to try to reduce storage, it was necessary to restrict the geometric shape of the boundary elements. In MINE3D, the element geometries are restricted to parallelepiped shapes. The parallelepipeds are contained within a three-dimensional grid in an i,j,k-coordinate system. Any interaction is then defined via the difference in the i,j,k-location between the two points of concern. In this system, there are

$$[(2*i_{max} - 1) \bullet (2*j_{max} - 1) \bullet (2*k_{max} - 1)]$$

possible different interactions with any node for a parallelepiped grid containing i_{max} nodes in the i-direction, j_{max} nodes in the j-direction, and k_{max} nodes in the k-direction.

This assumes that nodes are equally spaced in each of the three directions, although spacing in one direction can differ from that in another. Also, angles between the three directions are arbitrary. An example parallelepiped grid is shown in Fig. 2.

The coefficient storage requirements for the parallelepiped geometry are a linear function of the total number of nodes within the grid; for MINE3D, the total storage is roughly

$$6000 * i_{max} * j_{max} * k_{max}$$

It should be pointed out that all coefficients are calculated for a "box" enclosing all surface and volume elements. Thus, (i_{max} * j_{max} * k_{max}) is not directly related to N_e and N_p for the arbitrary-shape elements. Nevertheless, a gain in 3-D problem size is clearly achieved by using the parallelepiped grid. This is evidenced by comparing the size of the coefficient matrix for the arbitrary-shape elements to that for the parallelepiped grid, as shown in Fig. 3.

Even with the parallelepiped scheme, the storage requirements are high. Three-dimensional problems in bulk mining can be expected to require models with 10,000 and more surface and volume nodes; thus, storage requirements can easily exceed 200 Mbytes.

Unfortunately, computer hardware systems typically available to mining organizations are not capable of providing sufficient memory for array storage without a substantial degradation in

performance rate for disk access. Figure 4 presents a comparison of MINE3D run times (in terms of iteration rate) for three computers: (1) an IBM-compatible 80386 (25 MHz) microcomputer with Weitek coprocessor, 16 Mbytes memory and 120 Mbyte drive; (2) a SUN SPARCStation 1+ with 8 Mbytes memory and 200 Mbyte drive; and (3) a SUN SPARCStation 2 with 16 Mbytes memory and 860 Mbytes drive. As shown, the degradation of performance is dramatic for all three systems. The results infer that performance is more heavily dependent on RAM size than on machine type. At present, the limit for practical solution of problems with MINE3D is roughly 2000 surface nodes using the SUN SPARCStation 2. (See Volume 2 for a more detailed discussion.)

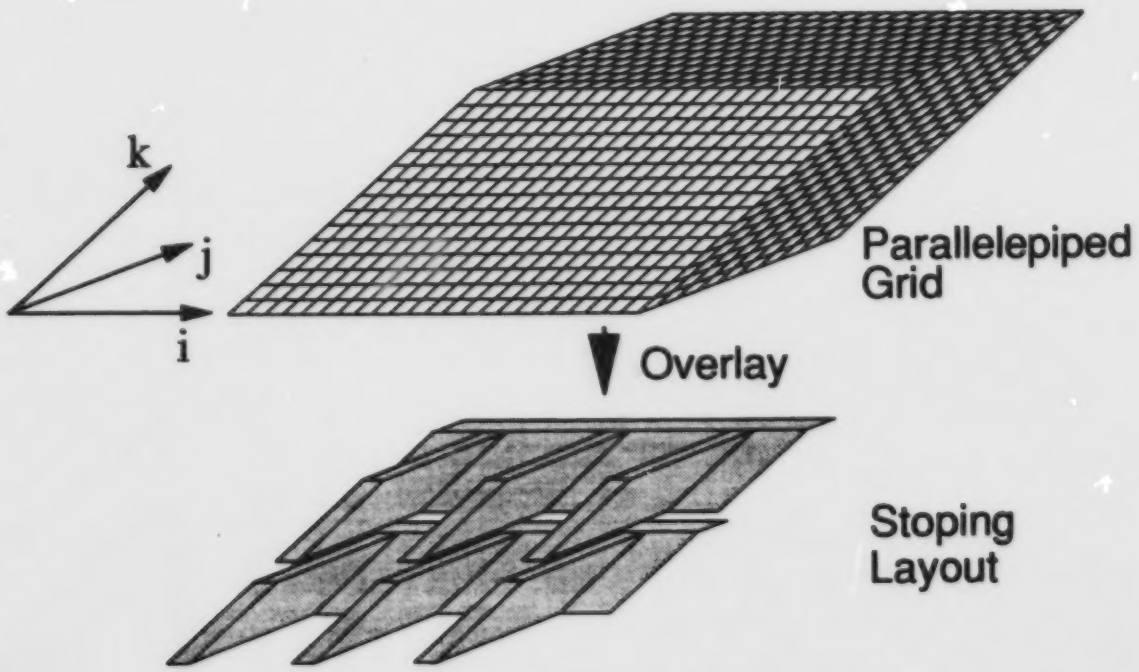


Fig. 2 A Parallelepiped Grid Overlaid On a Stoping Layout

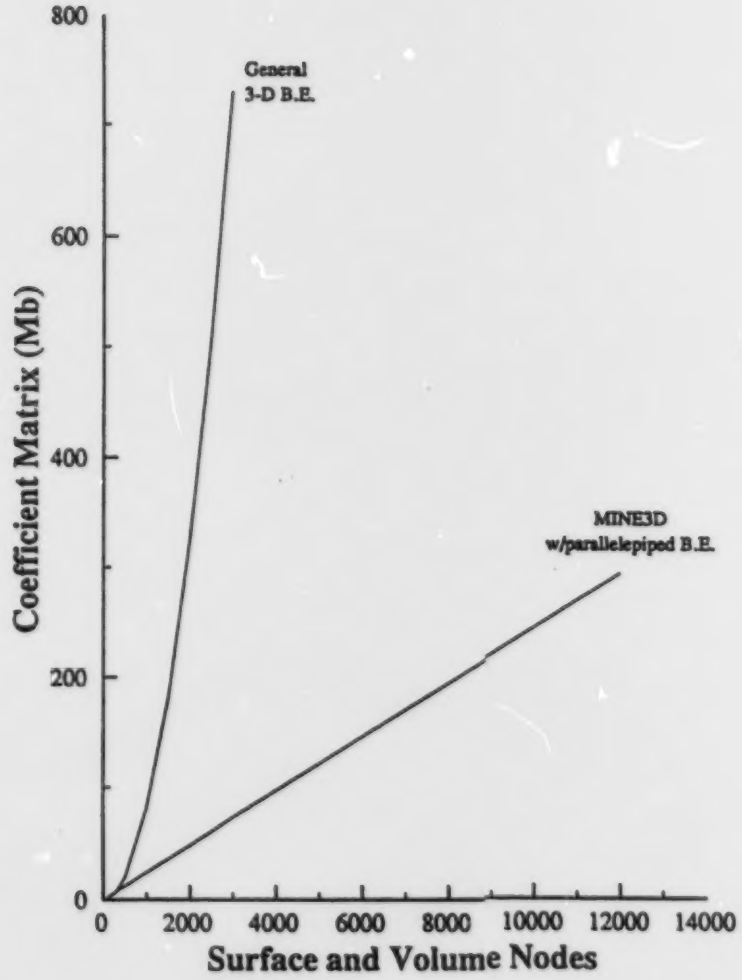


Fig. 3 ***Coefficient Matrix Size versus Number of Boundary Element Nodes***

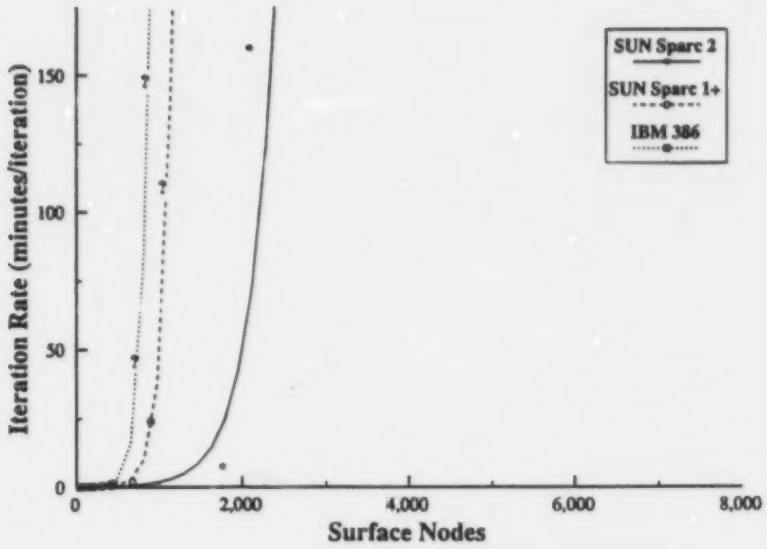


Fig. 4 ***Comparison of MINE3D Iteration Rate versus Number of Surface Nodes for Different Computer Systems***

3.0 NUMERICAL MODELING WITH MINE3D

3.1 *Software Overview*

MINE3D is written in FORTRAN77 and developed on an 80386-based microcomputer running DOS 4.x and using SVS FORTRAN 386, the PHARLAP linker and DOS extender. The program is divided into three separate codes: (1) MIP (MINE3D Input Processor), (2) MAP (MINE3D Analysis Program), and MOP (MINE3D Output Processor). MAP was also developed to run on a SUN SPARCStation environment running UNIX. The source files for the three codes are listed and described separately in Appendix I.

MIP and MOP are written specifically for operation on an 80386-based microcomputer with either 80387 or Weitek coprocessor. The MIP program requires a VGA card and monitor to display the menu graphics developed especially for this project.

The computational program MAP requires a considerable amount of computer memory and disk space in order to form the coefficient arrays needed for the 3-D boundary element solution, as discussed previously in Section 2.6. Thus, the code is written to be easily transportable to larger computers. The input file to MAP and output files from MAP are all ASCII formatted for direct transfer between computers.

Five types of files are generated for every model analysis. A MIP workspace file (extension .MIP) which the user develops when generating the MINE3D model, contains all data defining the grid, model conditions and mining steps plus AutoCAD data. An AutoCAD "DXF" file is read into MIP to use in model generation. Input data files with user-defined filenames (no extension assumed) are created by MIP for transfer to MAP. These data files can also be sent directly to MOP for making three-dimensional perspective plots. MAP creates three types of output files: a printed output along selected planes in the grid (extension .OUT), a graphics results file which is read by MOP (extension .GPR), and a restart file (extension .SAV). The files are discussed in more detail in Sections 4.0, 5.0 and 6.0.

3.2 *Program Requirements and Limitations*

The pre-processor (MIP) and post-processor (MOP) are designed to display large parallelepiped grids using relatively low computer memory requirements. Roughly 10,000 node grids can be viewed on a computer with 4 Mbytes RAM. An example plot is shown in Fig. 5. In MIP, the size of the main array is adjusted automatically to use all available computer memory. In MOP, the parameter NDIM, located in file "COMMON.BEM", must be increased and the code recompiled to increase the model size for more core memory.

Coefficient array storage and efficiency of disk IO are limiting factors in program MAP to the size of the model which can be analyzed. Presently, a model containing up to roughly 2000 surface and volume nodes is the practical limit. The parallelepiped grid size limitation is roughly 10,000 nodes.

The array sizes in MAP are defined in terms of the parameters IJKMAX and IVNMAX, located in file "BEMCOM". If the number of nodes exceed the pre-set values for these parameters, a message will be printed to the ".OUT" file, with the number to which IJKMAX or IVNMAX should be set. The storage requirement can be estimated from the following relation

$$\text{memory required to run MAP} \approx 0.04 + 0.024\ (i_{max} * j_{max} * k_{max})\ \text{[Mbytes]}$$

The user should be careful not to create a model grid with MIP which is too big for MAP to analyze with the available computer memory and disk space.

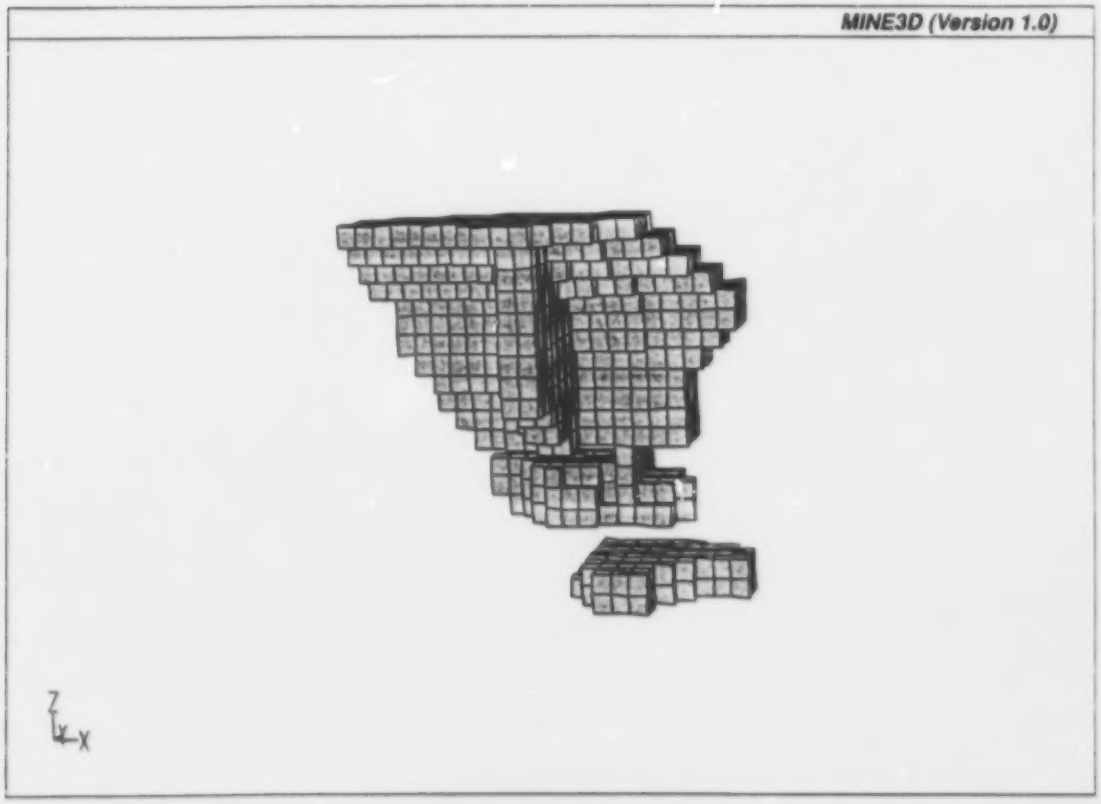


Fig. 5 ***Example 3-D Perspective Plot from MOP for a Model Containing 9261 Parallelepiped Nodes and 1748 Surface Nodes***

3.3 *Modeling Sequence*

Problem analysis with MINE3D is divided into three stages:

(1) Generate Input Data File(s) with MIP — The model is created by following the menu graphics steps described in Section 4.0. All conditions and parameters, for each mining step which is to be analyzed, must be defined before a data file is created. Several data files can be created to evaluate different parameter variations. A data file should be read into MOP before the analysis is performed in order to view the 3-D perspective plot and ensure the model geometry is correct at each mining step.

(2) Perform Boundary Element Solution with MAP — The data file is now read into MAP, the coefficient arrays are assembled, and the iterative solution proceeds. The input parameters are described in Section 5.0. The arrays are assembled once, at the beginning of the run. The iterative solution process is repeated for every mining step. The total number of iterations per mining step will vary depending on model conditions. Typically, the number of plastic iterations (and the number of elastic passes per plastic iteration) will be less than 100. The iteration count and solution error are printed to the screen so that the user may monitor the iteration rate.

(3) Evaluate Printed and Graphics Results — Printed output can be generated along user-selected planes. The output data include model coordinates, displacements, stresses, region numbers and plasticity indicators for all nodes on the selected plane.

The graphics output file (".GPR") is read into MOP, and results along any selected i, j or k plane through the model can be displayed. The graphics options are described in Section 6.0. Plots of stress contours, displacement vectors and plasticity state can be viewed. The graphics file can also be read back into MIP so that a comparison can be made between the calculated plasticity state and the user-selected plasticity region. If the calculated plasticity extends to the boundary of the selected plastic volume elements, then more elements should be added to ensure that plastic failure does not extend further.

3.4 *Save / Restart Procedure*

A save file (".SAV") is generated at the end of each MAP run. This file may be used to restart a second phase of an analysis at the last mining step without having to rerun the entire analysis from the beginning.

In order to restart the run, the ".SAV", ".GPR" and ".DAT" files are required from the first analysis phase. The ".SAV" and ".GPR" files from the first phase must be renamed to correspond to the name of the ".DAT" file for the second phase. The following example illustrates the save / restart procedure.

There are three data files:

> "TEST1.DAT" — This file runs mining step 1.
>
> "TEST2A.DAT" — This file includes input instructions for mining step 1 ***and*** mining step 2. The start mining step is defined as 2.
>
> "TEST2B.DAT" — This file includes input instructions for mining step 1 and an alternative mining step 2. Again, the start mining step is defined as 2.

MAP is first run with "TEST1.DAT" and, at completion, the following three files are created: "TEST1.OUT", "TEST1.GPR" and "TEST1.SAV". These files should then be copied to "TEST2A.OUT", "TEST2A.GPR" and "TEST2A.SAV". Now, MAP is run with "TEST2A.DAT". MAP will append "TEST2A.OUT" and "TEST2A.GPR" to the results from the second mining step. The file "TEST2A.SAV" will then correspond to the end of the ***second*** mining step.

For an alternative mining step, files "TEST1.OUT", "TEST1.GPR" and "TEST1.SAV" are copied to "TEST2B.OUT", "TEST2B.GPR" and "TEST2B.SAV". "TEST2B.DAT" is run in the same manner as "TEST2A.DAT". In order to continue to a third mining step, the files must be renamed, as above, and a new ".DAT" file created.

3.5 *Energy Calculation*

At the completion of each elastic mining step, the various energy components as described by Salamon (1984) are determined if requested in MIP. These components include:

(1) U_m — the stored strain energy content of the excavated volume, V_m, in the previous mining state;

(2) W_s — the energy expended in compression of backfill;

(3) W_r — the released energy; and

(4) W_k — the potential kinetic energy.

For problems using units of meters and Megapascals, the energy will be in Megajoules. These can also be expressed in volumetric rates as MJ/m^3 by dividing by the mined volume at any mining step. The values are printed to the output file (".OUT") at the completion of the print-out for all mining steps.

The calculation of energy dissipated through plastic deformation requires evaluation of intermediate values of stress increment from fictitious tractions during plastic iterations. These calculations increase the time for a run dramatically and, considering the limited usefulness of the information, is probably unwarranted, particularly in large runs. Therefore, in the present version of MINE3D, the energy calculations are computed the same way for both elastic and plastic runs. Caution should be used in interpretation of the energy dissipated in plastic deformations.

3.6 *Sign Conventions and Units*

The following sign conventions are used in MINE3D and must be kept in mind when entering input.

The sign convention for stresses in the boundary element solution scheme in MAP is as follows: the stress component σ_{ij} is positive if it acts in the ***positive*** (***negative***) j-direction on a plane whose outward normal points in the ***positive*** (***negative***) i-direction. This convention results in positive values for tensile stresses normal to the plane of action. The signs of the stresses tangent to the plane of action are chosen in a way that allows for a description by the same definition used for stresses normal to the plane. This sign convention is generally accepted for mathematical formulations in elasticity and plasticity.

In mining engineering, a common convention is that compressive stresses are positive. For this reason, an option is available in MIP to allow the user to select a sign convention for input and output stresses. The default sign convention is that compressive stresses are positive. The stress components for the default convention for input and output are shown in Fig. 6.

The sign conventions for strain and displacement are: (a) extensional strain is positive; and (b) displacement is positive in the direction of the positive x-, y-, or z-axes.

All material properties (including tensile strength) are input as positive values.

SI units are recommended; however, any consistent set of engineering units for properties, stresses and dimensions can be used. A review of systems of units is given in Table 1. No conversions are performed in the program except for friction and dilation angles, which are entered in degrees. For plotting purposes, it is best to keep units of stress smaller than 10^4.

3.7 *Hard-Copy Plotting*

Hard-copy plots can be created from the graphics screen plots displayed in MOP. Plots are generated for a PostScript$^{(R)}$ laser printer and can be sent to a disk file for storage or later copying to the printer. A post-processor is presently under development to send PostScript-formatted files to an HP Paint Jet printer to produce color plots. This software is not yet completely tested. Example color plots are attached in Appendix II.

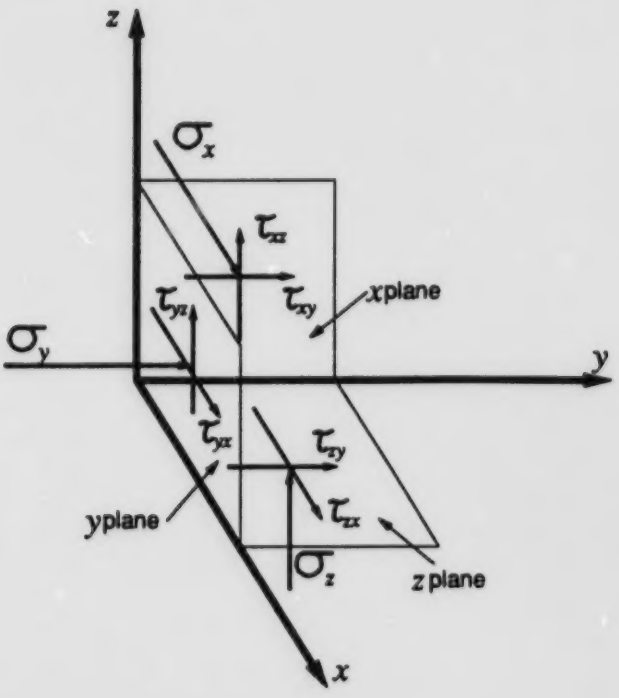


Fig. 6 Default Sign Convention for Input and Output Stresses in MINE3D (Note that the outward normal for the x-, y-, and z-planes points in the negative x-, y-, and z-directions.)

Table 1
SYSTEMS OF UNITS

	METRIC				BRITISH	
Length	m	m	m	cm	ft	in
Density	kg/m^3	10^3 kg/m^3	10^6 kg/m^3	10^6 g/cm^3	$slugs/ft^3$	$snails/in^3$
Force	N	kN	MN	Mdynes	lb_f	lb_f
Stress	Pa	kPa	MPa	bar	lb_f/ft^2	psi
Gravity	m/sec^2	m/sec^2	m/sec^2	cm/s^2	ft/sec^2	in/sec^2
Stiffness	Pa/m	KPa/m	MPa/m	bar/cm	lb_f/ft^3	lb/in^3

where 1 bar = 10^6 dynes/cm^2 = 10^5 N/m^2 = 10^5 Pa,

1 atm = 1.013 bars = 14.7 psi = 2116 lb_f/ft^2 = 1.01325x10^5 Pa,

1 slug = 1 lb_f - s^2/ft = 14.59 kg,

1 snail = 1 lb_f -s^2/ in, and

1 gravity = 9.81 m/s^2 = 981 cm/s^2 = 32.17 ft/s^2.

4.0 MINE3D INPUT PROCESSOR (MIP)

MIP is a graphical input processor for MAP that runs on an 80386-type personal computer with a VGA screen. It creates an input file for the solution program MAP that specifies the geometry, properties, stresses, mining steps, plasticity regions and solution parameters. Since the program is menu-driven, very little is required by way of a user's manual. A simplified flow diagram for MIP's menu tree is shown in Fig. 7. On many of the screen displays, HELP boxes are available, but these are omitted from Fig. 7, for clarity.

The mouse is the most convenient way to select items on the menu screens, but the arrow keys on the PC keyboard may also be used to move the cursor around. The left-hand mouse key or ENTER key is used to pick an item that has been selected by the cursor.

The main function of MIP is to allow the user to specify mining patterns in three dimensions simply by pointing to the appropriate elements of a grid with the mouse. MIP can also display geometrical information derived from the AutoCAD program. The AutoCAD information is not used directly to create mining patterns in the numerical grid; rather, the user matches his mining pattern to the AutoCAD drawing, which appears as an overlay. This matching process is quite easy to do, and is certainly easier than specifying boundary element data in numerical form. Inexperienced users can use MIP to produce a MAP input file in a matter of minutes.

4.1 *Initial Entry to MIP*

On entry to MIP, the user can start a new run, or continue a run that was previously saved: box 1 on Fig. 7 indicates this choice. For a new run, the dimensions of the boundary element grid must be given, together with the number of elements in the three coordinate directions. Optionally, the origin and angles of axes may be given; if these are not given, the origin (lowest X,Y,Z value) defaults to (0,0,0) and the axes default to a rectangular set, aligned with the global (X,Y,Z) axes. Non-rectangular axes for parallelepipeds are specified in terms of their dips and dip directions with respect to the global axes. A right-handed coordinate set should be used — conventionally, X corresponds to East, Y corresponds to North and Z corresponds to Up (i.e., Z decreases, or goes negative, with depth). MIP checks the given grid data for consistency, and then allows the user to pass to the main menu (box 3) if everything is reasonable. In this case, a 2-D slice through the grid is displayed. If MIP rejects the grid, the user is invited to check the given data.

4.2 *MIP's Main Menu*

When the main menu is displayed, the user is able to change elements in the grid by placing the cursor over an element and touching the left-hand mouse button. The element is then changed to the selected type, and a new color displayed for the element. By default, the element type will be "mined out," which is shown as black, but the selection may be changed by going to menu box 4. Here, a new material number may be selected for subsequent picks with the cursor. Alternatively, the top box ("P") allows the user to specify elements that can go plastic in the MAP run. The "P" flag may be turned off or on in any element when the main-menu screen is active.

Brackets < > in the menu boxes of Fig. 7 denote that the enclosed items may or may not appear, depending on circumstances.

Upon first entry to the main menu, a 2-D slice through the grid will be displayed, parallel to the X′-Y′ axis (the primes denote the parallelepiped axes, which may be at angles to the global axes). New slices (parallel to the current one) may be displayed by picking menu item **down one slice** or **up one slice**. The words "up" and "down" denote an increase or decrease in slice number, not necessarily a movement up or down in real space. Slices parallel to the other two coordinate directions may be selected by picking the items **show Y′-Z′ plane** or **show Z′-X′ plane**. It is possible to jump directly to a given slice (or step — see Section 4.2.1) by selecting the **Jump to** ... item: this is useful for large grids. The functions **zoom in**, **zoom out** and **pan** enable the picture to be magnified, diminished and shifted, respectively. When **pan** is selected, movement of the mouse controls the movement of the outline of a new viewing window and its center (denoted by a cross). Pressing the left mouse button then fixes the viewing window at its new location.

Patterns of mining and filling may be copied between different slices by picking the **Copy pattern** item. By default, the pattern in the current slice is copied to the next slice, but the user may type in any valid slice(s) under the FROM-TO headings.

The white message in the left-hand corner of the viewing window gives the current axes and slice number selected, together with the mean coordinate of the slice in the direction of the remaining axis. Also given is the **STEP NUMBER**: this corresponds to the mining step that is currently being constructed. The **x** and **y** numbers in the corners of the window denote the coordinates of those corners. If it is required to know the coordinates of a particular object on the screen, then the **pan** function can be used to position a window edge or corner on the object — the coordinate(s) can then be noted.

4.2.1 Mining Steps

When the next mining step is selected (using the **next step** menu item), a completely new 3-D grid is provided. Initially, all the data in the previous grid are copied to the new grid, since it is anticipated that new mining patterns will consist of modifications to previous patterns, rather than completely new ones. The user may edit the new grid for the new mining step, and then request a further step if necessary. It is also possible to return to previous steps, to check or modify data.

WARNING! Every pick of the **next step** menu item will create an additional grid that will cause MAP to execute an additional calculation step; be sure that a new step is really intended before picking the item.

4.2.2 Print Request

The blue box labeled **print request** in the top, right-hand corner of the window may be picked if grid printout from MAP is required for the slice (and mining step) that is currently displayed. The item may be turned on or off (white letters denote 'on' and black letters denote 'off').

4.2.3 Reading and Writing Files

Menu box 5 is activated when the main menu item **file functions** is picked. From this menu, the workspace may be saved and restored. The term "workspace" means: all grids and steps currently defined, together with data such as properties, stresses and options. Also saved is any AutoCAD data that have been input to MIP. The workspace file is automatically given the extension **.MIP**.

The **get AutoCAD file** item is used to read a "DXF" file that has been created by the AutoCAD program. The entities recognized are POLYLINES and LINES, but no error will occur if other entities are present in the DXF file. Once the DXF file has been read by MIP (which may take a considerable time), the data is stored in compact form, and need not be read in again if a workspace is saved and then restored.

Once the grid has been created, together with all the desired mining steps, a MAP input file can be created by picking the **write MAP file** menu item. This creates an ASCII file with a filename given by the user (no extension is assumed). Since the file is in ASCII format, it can be edited by the user if necessary.

The menu item **get plas.file** enables a set of plastic points computed by MAP to be displayed by MIP, in order to guide the future selection of plastic elements. The computed plastic nodal points are displayed with red 'P' symbols. When prompted, the appropriate filename (usually a .GPR type file created by MAP) should be given. If no file name is specified, all previously-displayed red Ps will be erased.

4.2.4 Properties, Parameter and Options

Picking the item **properties, etc.** on the main menu causes a new menu to appear. Material properties may be given for the 7 material types that are currently allowed. Note that material number 7 corresponds to the virgin rock. When a connection between two materials exists, MIP chooses boundary conditions for MAP based on the Young's modulus for the two materials. Consequently, fill material should be assigned a lower modulus than the virgin rock, in order to assure correct operation of MAP. Stresses may also be defined for up to 10 "regions" in the grid. The virgin rock corresponds to region number 1; other region numbers are allocated sequentially to contiguous regions of different material number. A problem heading (which is transmitted to MAP) may be given by picking the **problem heading** item. The **solution parameters** menu item allows selected operating parameters for MAP to be changed — see Section 5.0 for information. Finally, the **options, settings** item causes a sub-menu to be displayed, in which one of three AutoCAD display modes may be selected (see Section 4.2.5 for information). The items in the sub-menu (box 11 of Fig. 7) are switched on or off by clicking on them with the mouse; only one may be active at a time. The mode **All CAD data visible** must be turned off before either of the other two can be selected.

4.2.5 AutoCAD Layers

The information contained in an AutoCAD "DXF" file may be organized into "layers," which are defined by the person who created the AutoCAD drawing. "Layers" are used to divide a complex drawing into a series of named collections of items; layers do not necessarily correspond to physical divisions of space. It is possible to switch any layer on or off (i.e., displayed or not displayed), by picking the item **show A-CAD layers** from the main menu. The subsequent screen shows all the layers present — they may be turned on or off by picking them with the cursor and touching the left mouse button. There are three options that control how AutoCAD information is displayed:

1. **All lines displayed.** In this mode, all POLYLINES and LINES in the active (i.e., switched on) layers are displayed.

2. **Bracketed data.** Only display information that lies within the height of the current slice is displayed; in this mode, the AutoCAD drawing will disappear if there are no active layers within the selected slice.

3. **Nearest data.** In this mode, display data are always visible (if some active layers exist) — the nearest lines are displayed, provided that they fall within a window that has the height on one slice.

The three modes are selected by a sub-menu called from the **property, data, etc.** menu (number 6 on Fig. 7).

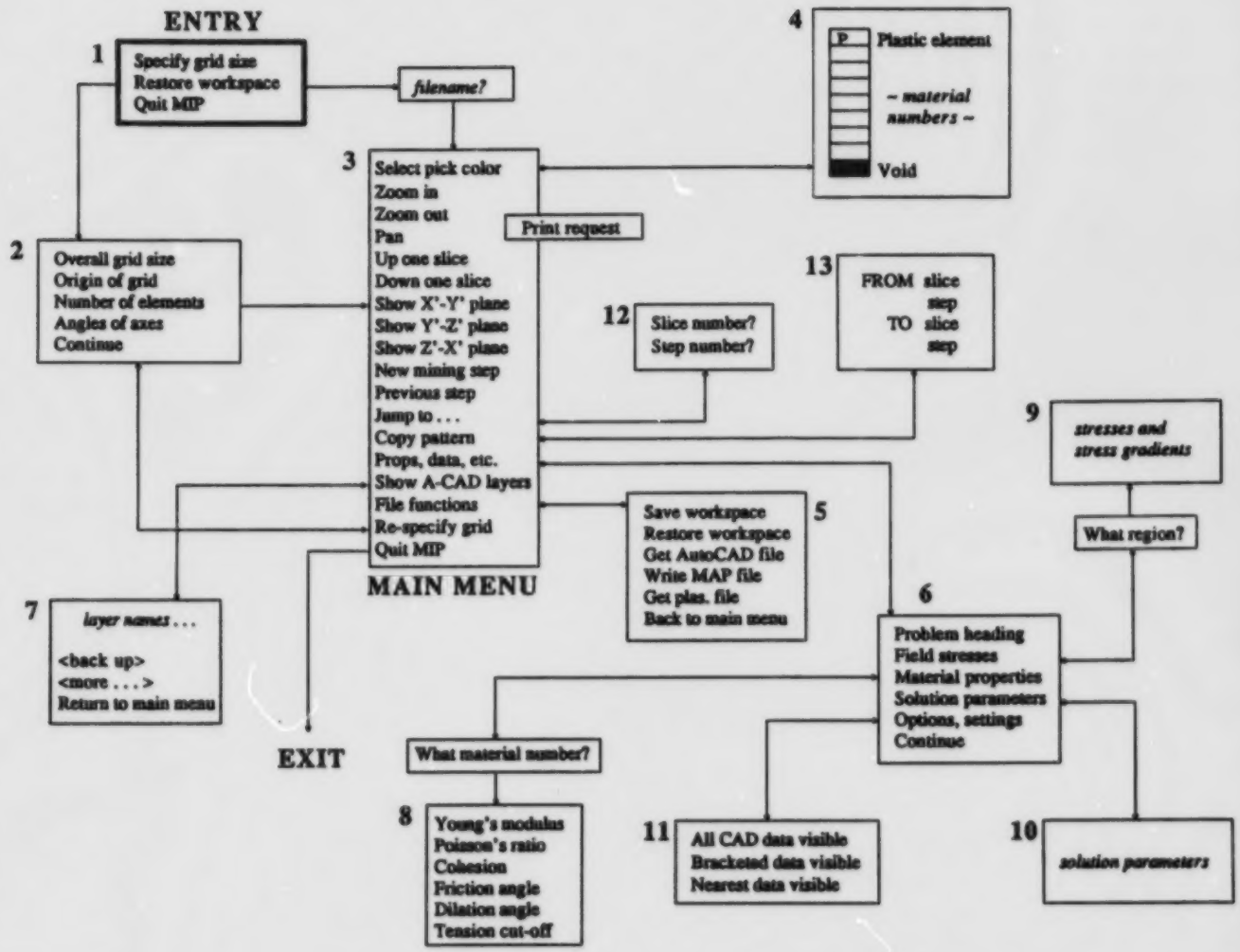


Fig. 7 Menu Tree for MIP

5.0 MINE3D ANALYSIS PROGRAM (MAP)

MAP performs all the boundary element calculations using data files generated by MIP and creates either printed output or graphics output files which can be sent to MOP for viewing. MAP can be run on a separate computer system from MIP and MOP since both input and output files are written in ASCII format. The input data file generated by MIP may be edited manually — for example, if a series of analyses with different material properties or far-field stress conditions are desired. However, if the user wishes to make changes to the geometry or mining sequence, this is best done by modifying the workspace file and creating different data files in MIP.

5.1 *Element and Node Definitions*

The data file required for MAP is primarily a list of integers and integer sets which identify elements and nodes, turns them on or off at different mining steps and connects adjacent nodes of different regions of materials.

The definitions for element and node locations in the parallelepiped grid are illustrated in Figs. 8 to 10. Surface nodes are identified by a set of three i,j,k integers, as shown in Fig. 8. Surface element planes are identified by an i,j,k integer location, offset midway between nodes, and a plane "set number". There are three surface element planes associated with a surface element location, as shown in Fig. 9. The set numbers identify each plane: set 1 is the j-k plane, set 2 is the i-k plane, and set 3 is the i-j plane. The set number also has a positive or negative sign indicating the direction of the outward normal from the surface element plane. Plastic volume elements and volume nodes are identified by i,j,k integer sets as shown in Fig. 10.

In addition to node and element locations, the data file also provides an integer code identifying the normal direction to surface nodes and volume nodes. The normal depends on the number and combination of parallelepiped volumes surrounding the node. Each node has a maximum of eight possible volumes surrounding it. Thus, there are 2^8, or 256, combinations of volumes surrounding a node. For a surface node, some combinations of volumes must be removed, but at least one volume must be removed and at least one must remain. This leaves $2^8 - 2 = 254$ possible combinations of volumes. In MINE3D, each volume is assigned a code, as defined in Fig. 11. The nodal code is then defined as the sum of all volumes present. Each code corresponds to a unique combination of volumes present. For example, if volume i,j-1,k is missing, the node code is 255 - 32 = 223 (see Fig. 11). The normal to this node is thus uniquely defined by the sum of volumes present around the node. The node codes for the 8 surface nodes comprising a square void are shown in Fig. 12.

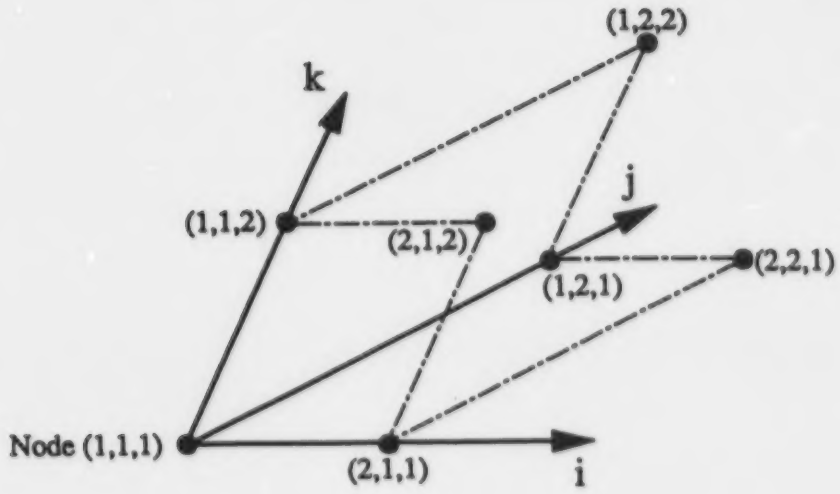


Fig. 8 Surface Node Definition

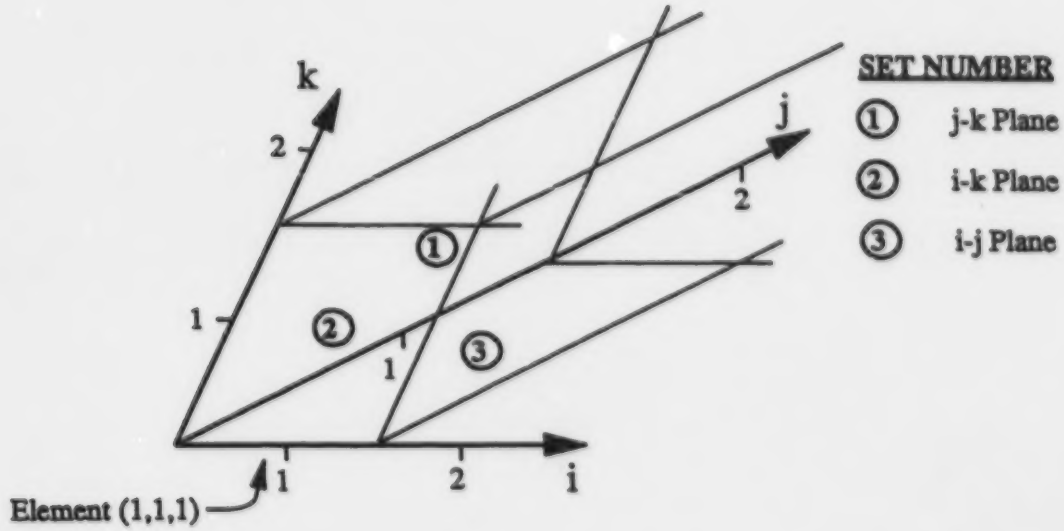


Fig. 9 Surface Element Definition

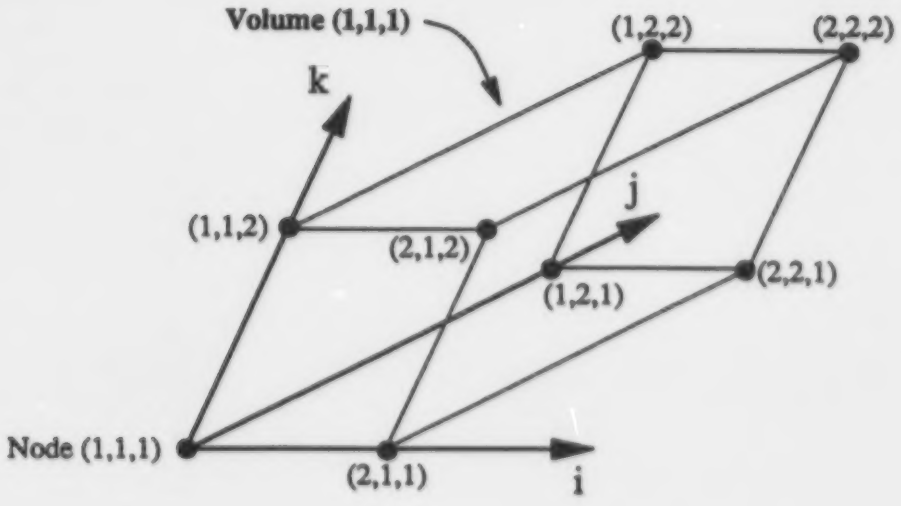


Fig. 10 Volume Element / Node Definition

Volume Element	Code
i-1, j-1, k-1	1
i-1, j-1, k	2
i-1, j, k-1	4
i-1, j, k	8
i, j-1, k-1	16
i, j-1, k	32
i, j, k-1	64
i, j, k	128
	255

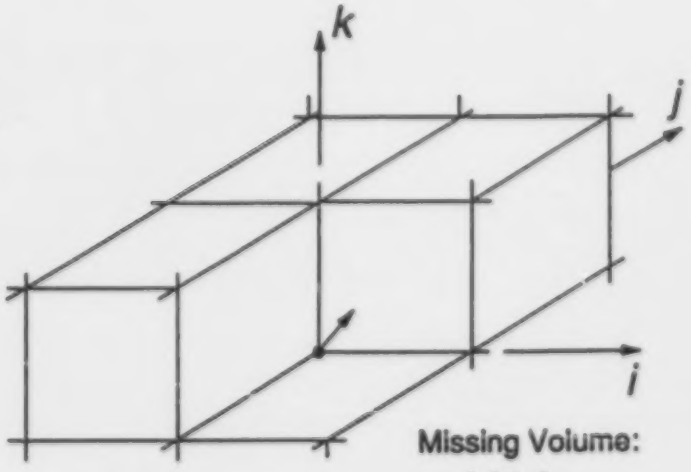
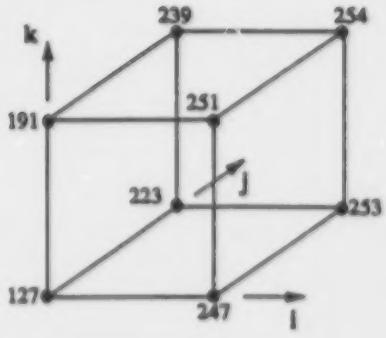


Fig. 11 Integer Code for Nodal Coefficients

NODE CODE	MISSING VOLUME
127	i,j,k
191	i,j,k-1
223	i,j-1,k
239	i,j-1,k-1
247	i-1,j,k
251	i-1,j,k-1
253	i-1,j-1,k
254	i-1,j-1,k-1



6 Surface Elements

Fig. 12 Node Codes for a Square Void

Plastic volume nodes have the same code, except that, now, 255 combinations are possible because a volume node can exist which is entirely surrounded by parallelepiped volumes.

For multiple materials, an integer list is also given which identifies which surface nodes of any two materials in contact are located at the same global position. These nodes are then defined as twin nodes for applying the appropriate displacement or traction boundary condition.

Initially, all nodes are assigned region number 1 and material number 7. When elements are excavated, nodes falling within the excavated or void region are changed to region number 0. If voids are filled with a different material, new region numbers are assigned automatically to these regions by MIP. Different far-field stress states can be assigned to separate regions. Different regions can have the same material number but must ***not*** have the same region number.

WARNING: If a void between two separated regions is filled, thus connecting the regions, the material number ***must*** be the same for the regions.

The input data file for MAP consists of parameter sets which define the parallelepiped grid, solution parameters, far-field stress state for each region, material properties for each material, and lists of all surface elements and nodes and volume element and nodes in the model. Parameter sets are also given for all elements and nodes turned on (positive sign) or turned off (negative sign) and connected (positive sign) or disconnected (negative sign) at each mining step.

5.2 *Input Data File*

The input data file is grouped into 18 parameter sets, as shown in Table 2. The first ten sets define the initial parameters for the model. The remaining eight sets define the conditions at a given mining step. These sets are repeated for every mining step in a problem. All parameter sets, except for sets 1, 2, 3 and 4 (as defined in Table 2), are separated from the other sets by the integer zero.

5.2.1 Problem Heading

This heading is printed on all printed output files and graphics plots.

5.2.2 Solution Parameters

A set of 10 solution parameters must be defined.

MSTEP is the total number of mining steps defined for the model.

NIT is the maximum number of iterations allowed for solution at each step. **NIT** must equal 1 if an elastic-only analysis is run. **NIT** will typically equal 100 if a plasticity analysis is performed.

NITE defines the maximum number of elastic passes per iteration step. **NITE** is typically set equal to 100.

NITP is set equal to zero if an elastic-only analysis is run and set equal to 1 for a plastic analysis.

RPMIN is the successive overrelaxation relaxation parameter for the iterative solution technique. This value is typically set equal to 0.80.

EPFAC is the corner offset fraction. This fraction is the ratio of the offset distance to the smallest characteristic length of the elements. This value is set typically to 0.01. However, it has been found that for models with reentrant corners, **EPFAC** must be reduced.

TOL is the plastic stress tolerance (or fictitious force tolerance for elastic problems) for stopping the iterative solution. **TOL** is typically set to approximately 0.001 times the minimum far-field stress and the value should be decreased for improved accuracy.

NERG is set equal to zero if no energy calculation is to be made, and set equal to 1 if an energy calculation is to be made.

NSIGN is the sign convention for stresses; **NSIGN=0** (default) if compressive stresses are positive, and **NSIGN=1** if tensile stresses are positive.

MSTART is the mining step number to start the solution. The default is mining step number 1.

5.2.3 Parallelepiped Nodes

This line contains the total number of parallelepiped nodes in the i-direction (i_{max}), j-direction (j_{max}) and k-direction (k_{max}). Also given are the total number of surface elements (nse), the total number of surface nodes (nsn), the total number of volume elements (nve), the total number of volume nodes (nvn), and the total number of connected nodes (ncnn) in the model for all mining steps. It will be necessary to increase the size of storage arrays in MAP if the total number of nodes and elements exceeds a pre-set limit.

5.2.4 Global Coordinates for Parallelepiped Grid

The global coordinates of four corners of the parallelepiped grid are specified. These coordinates correspond to locations: (i=j=k=0), (i=imax, j=k=0), (j=jmax, i=k=0), and (k=kmax, i=j=0).

5.2.5 Far-Field Stress Parameters for Each Region

Each region in the model is automatically assigned a region number by MIP. The infinite body region is always region number 1. Excavations or voids in the model are assigned region number zero. When fill material is added to a void, a new region number is assigned (beginning with 2). An initial far-field stress state is assigned to each region number greater than or equal to 1. The six components of the global stress tensor, plus gradients, can be given. Stresses in the model are calculated from the relation:

$$\sigma_{ij} = fs_{ij} + fsg_{ij} * d \qquad (i \text{ and } j = x,\ y \text{ or } z) \tag{8}$$

where σ_{ij} = the stress components,

fs_{ij} = the far-field stress parameter,

fsg_{ij} = the vertical gradient in the far-field stress component (e.g., MPa/m), and

d = the vertical distance measured from the parallelepiped grid corner (i=j=k=0).

5.2.6 Material Properties for Each Material

Each different material in the model is assigned a material number. The infinite body material number ***must*** be set equal to 7. For each material number assigned in the model, a set of material properties must be given. These properties are Young's modulus (E), Poisson's ratio (ν), cohesion (c), friction angle (ϕ), dilation angle (ψ), and tensile strength (T). E and ν must be given for every material. If any other parameters are not specified, the default value is zero. ν ***must*** be equal for all materials.

5.2.7 Surface Element List

A sequential list is given of every surface element assigned in the model for all mining steps. This list specifies the i-, j-, k-integer coordinates, set number (iset), material number (mat), and region number (ireg) for each surface element (see Fig. 9).

5.2.8 Surface Node List

A sequential list is given of every surface node assigned in the model for all mining steps. This list specifies the i-, j-, k-integer coordinates, the normal code number (1 through 254), the boundary condition (=1 for stress boundary, =2 for displacement boundary), the material numbers (mat) and the region number (ireg) for each surface node (see Fig. 8).

5.2.9 Volume Element List

A sequential list is given of every volume element assigned in the model for all mining steps. The list specifies the i-, j-, k-integer coordinates, the material number (mat), and region number (ireg) for each volume element (see Fig. 10).

5.2.10 Volume Node List

A sequential list is given of every volume node assigned in the model for all mining steps. The list specifies the i-, j-, k-integer coordinates, the node normal code (1 through 255), the material number (mat), and the region number (ireg) for each volume node (see Fig. 10).

The remaining parameter sets are repeated for every mining step.

5.2.11 List of Surface Elements Turned On or Off for This Mining Step

The numbers of all new surface elements used (turned on) during this mining step are listed with a positive sign. If a surface element used at the previous mining step is still used, it is not repeated. If a surface element used at the previous step is now not used, it is turned off (negative sign).

5.2.12 List of Surface Nodes Turned On or Off for This Mining Step

The numbers of all new surface nodes used (turned on) during this mining step are listed with a positive sign. If a surface node used at the previous mining step is still used, it is not repeated. If a surface node used at the previous step is now not used, it is turned off (negative sign).

In addition, the node normal code (norcod) and boundary condition (ibc) are given, since these could change at the new mining step.

5.2.13 List of Volume Elements Turned On or Off for This Mining Step

The numbers of all new volume elements used (turned on) during this mining step are listed with a positive sign. If a volume element used at the previous mining step is still used, it is not repeated. If a volume element used at the previous step is now not used, it is turned off (negative sign).

5.2.14 List of Volume Nodes Turned On or Off for This Mining Step

The numbers of all new volume nodes used (turned on) during this mining step are listed with a positive sign. If a volume node used at the previous mining step is still used, it is not repeated. If a volume node used at the previous step is now not used, it is turned off (negative sign).

In addition, the node normal code (incod) is given, since this could change for the new mining step.

5.2.15 List of Connected Nodes for This Mining Step

The twin node numbers are listed of the two surface nodes which are connected at this step. Also given are the material numbers (m1 and m2) and region numbers (ir1 and ir2) for the two nodes. If the first integer on the line has a negative sign, then the nodes are disconnected for this step. Only new connections are listed. Old connections still apply until disconnected.

5.2.16 List of Altered Nodes for This Mining Step

Any nodes within the parallelepiped grid are listed which have their region number changed for this mining step (e.g., by changing to a void region or filled region). The i-, j-, k-integer coordinate and new material and region numbers are given for any node altered for this mining step. Note that this includes any node in the parallelepiped grid.

5.2.17 Volume of Region Excavated for This Mining Step

MIP calculates the total ***new*** excavation volume for each mining step and lists this value in the data file. This value is used for energy release calculations.

5.2.18 Selected Output Plane

The user may select any i-, j-, k-plane in the parallelepiped grid to write the calculated stress, displacement and plastic state to the output file (".OUT").

Table 2
MAP INPUT DATA FILE

PARAMETER SET	DESCRIPTION
1	Problem Heading
2	Solution Parameters

MSTEP NIT NITE NITP RMPIN EPFAC TOL NERG NSIGN MSTART

MSTEP	number of mining steps
NIT	maximum number of iterations for solution
NITE	maximum number of elastic passes per iteration
NITP	plastic analysis? (0=no, 1=yes)
RPMIN	relaxation parameter
EDFAC	corner offset fraction
TOL	tolerance on iterative solution
NERG	energy calculation? (0=no, 1=yes)
NSIGN	sign convention (0=compression positive; 1=tension positive)
MSTART	start solution at this mining step number

3 Parallelepiped Nodes

imax jmax kmax nse nsn nve nvn ncnn

imax	number of nodes in i-direction
jmax	number of nodes in j-direction
kmax	number of nodes in k-direction
nse	total number of surface elements
nsn	total number of surface nodes
nve	total number of volume elements
nvn	total number of volume nodes
ncnn	total number of connected nodes

Table 2
(continued)

PARAMETER SET	DESCRIPTION
4	Global Coordinates for Parallelepiped Grid
Corner 1	x,y,z coordinates at i=j=k=0
Corner 2	x,y,z coordinates at i=imax, j=k=0
Corner 3	x,y,z coordinates at j=jmax, i=k=0
Corner 4	x,y,z coordinates at k=kmax, i=j=0
5	Far-Field Stress Parameters for Each Region

(Compressive stresses have positive sign, by default.)

nreg fsxx fsxxg fsyy fsyyg fszz fszzg

fsxy fsxyg fsxz fsxzg fsyz fsyzg

nreg	region number
fsxx	xx-stress
fsxxg	xx-stress gradient
fsyy	yy-stress
fsyyg	yy-stress gradient
fszz	zz-stress
fszzg	zz-stress gradient
fsxy	xy-stress
fsxyg	xy-stress gradient
fsxz	xz-stress
fsxzg	xz-stress gradient
fsyz	yz-stress
fsyzg	yz-stress gradient

NOTE: The infinite body region number must equal 1.

Linearly-varying far-field stresses, σ_{ij}, varying with vertical depth, d, measured from the origin (0,0,0), are set from the following equations:

$$\sigma_{xx} = \text{fsxx} + (\text{fsxxg})\ \text{d}$$
$$\sigma_{yy} = \text{fsyy} + (\text{fsyyg})\ \text{d}$$
$$\sigma_{zz} = \text{fszz} + (\text{fszzg})\ \text{d}$$
$$\sigma_{xy} = \text{fsxy} + (\text{fsxyg})\ \text{d}$$
$$\sigma_{xz} = \text{fsxz} + (\text{fsxzg})\ \text{d}$$
$$\sigma_{yz} = \text{fsyz} + (\text{fsyzg})\ \text{d}$$

Table 2
(continued)

PARAMETER SET	DESCRIPTION
6	Material Properties for Each Material (all properties have positive sign)

nmat E ν C ϕ ψ T

nmat	material number
E	Young's modulus
ν	Poisson's ratio
C	cohesion
ϕ	friction angle
ψ	dilation angle
T	tensile strength

NOTE: The infinite body material number must equal 7.

7 Surface Element List

i j k iset mat ireg iel

i,j,k	surface element integer coordinates
iset	set number 1 = j-k plane 2 = i-k plane 3 = i-j plane
mat	material number
ireg	region number
iel	surface element number

8 Surface Node List

i j k norcod ibc mat ireg inod

i,j,k	surface node integer coordinates
norcod	node normal code (1 through 254)
ibc	boundary condition 1 = stress boundary 2 = displacement boundary
mat	material number
ireg	region number
inod	surface node number

Table 2
(continued)

PARAMETER SET	DESCRIPTION
9	Volume Element List
	`i j k mat ireg iel`
	i,j,k — volume element integer coordinates mat — material number ireg — region number iel — volume element number
10	Volume Node List
	`i j k incod mat ireg`
	i,j,k — volume node integer coordinates incod — node normal code (1 through 255) mat — material number ireg — region number

Parameter sets 11 through 18 are repeated for every mining step.

11	List of Surface Elements Turned On or Off For This Mining Step
	`iel`
	iel — surface element number (+ = turned on, - = turned off)
12	List of Surface Nodes Turned On or Off For This Mining Step, Including Normal Code and Boundary Condition
	`inod norcod ibc`
	inod — surface node number (+ = turned on, - = turned off) norcod — node normal code (1 through 254) ibc — boundary condition 1 = stress boundary 2 = displacement boundary
13	List of Volume Elements Turned On or Off For This Mining Step
	`icl`
	icl — volume element number (+ = turned on, - = turned off)

Table 2
(continued)

PARAMETER SET	DESCRIPTION
14	List of Volume Nodes Turned On or Off For This Mining Step, Including Normal Code
	`inod incod`
	inod — volume node number (+ = turned on, - = turned off)
	incod — node normal code (1 through 255)
15	List of Connected Nodes For This Mining Step
	`n1 m1 ir1 n2 m2 ir2`
	n1 — first node in connection
	m1 — material number for first node
	ir1 — region number for first node
	n2 — second node in connection
	m2 — material number for second node
	ir2 — region number for second node
16	List of Altered Nodes For This Mining Step
	`i j k mat ireg`
	i,j,k — node coordinates for altered node
	mat — new material number
	ireg — new region number
17	Volume of Region Excavated for This Mining Step
	`vexc`
	vexc — excavation volume
18	Selected Output Plane
	`lp n`
	lp — selected plane (i, j or k)
	n — plane number

6.0 MINE3D OUTPUT PROCESSOR (MOP)

MOP is used to view the element geometries set up in MIP and the output calculated in MAP. MOP, as with MIP, also runs on an 80386-type personal computer with a VGA screen. The graphics presentation in MOP is fully three dimensional in perspective, as opposed to the plane-section approach of MIP. In MOP, the user may rotate and translate the model in any direction, zoom in on particular features, and plot planes of calculated stresses, plasticity or displacements. The MOP program has two windows: the first is a text-based menu window, and the second is the 3-D graphics window. Interaction with the program is via the keyboard, as MOP does not currently support a mouse. The input to MOP is a file from MIP which normally has a file extension ".DAT". MOP will also read the calculated data file from the MAP program if it is present. The calculated data file will have the same name as the ".DAT" file, but will have a ".GPR" extension If a ".GPR" file is not present, the user may still plot the 3-D view of the model, but may not plot calculated results.

When operating in either the menu or graphics window, the <esc> key will abort the entry on a particular line. So, if the user has accidentally started to enter a command, <esc> will back up to the start of that entry. The "F1" key is used to back up one level. If the user is in the graphics window, the "F1" key will cause a jump to the menu window.

6.1 *Menu Window*

The menu window is used to control plotting parameters and to direct reading of data files. Movement through the menu is via the up and down cursor keys. The "F1" key will always jump up to the first line and then, by pressing the ENTER key, the user enters the graphics window. At any time, pressing the <esc> key will abort data entry to the current line and restore the previous status. Movement above the top entry line will jump to the bottom, and movement past the bottom will jump to the top.

Menu Items

View Image

Pressing ENTER on this line will cause a jump to the graphics window.

Solid View

Pressing ENTER on this line will toggle through three possible plot states: wire-frame, solid with no interior surfaces, and solid with interior surfaces. The interior sides of elements are visible when the foreground is hidden and "solid with interior surfaces" is selected.

Outline

Pressing ENTER on this line will toggle between outline and non-outline element plots. Non-outline is legal only for solid plots.

Foreground

Pressing ENTER on this line will toggle the foreground control. Selecting the "hide" on foregrounds will prevent plotting any surface elements in front of an output data plot.

Color

Pressing ENTER on this line will toggle between color by material number or by region number.

Read MIP

On this line, the user must enter the name of the data file that is to be viewed. (The extension is not required.)

Current Mining Step

For data files with multiple mining steps, the user can specify the one to display.

Hard-Copy Filename

The user may enter the name of a file which will be sent PostScript data generated in the graphics window by pressing the "F10" key. The default for this is LPT1, which is the parallel printer port.

Object Center

The user may view or modify the center of rotation of the objects in the graphics window.

Hide Material

The user may select to hide elements of a specific material by entering a "1" on the line below the material number.

Hide Region

The user may select to hide elements of a specific region by entering a "1" on the line below the region number.

Stress Range

If a ".GPR" file is present, the stress range will be automatically set when the file is read. If the user enters new values into the stress range, this range will remain until a new file is read. (Changing mining steps will not change the stress range if entered by the user.)

Displacement Scale

If a ".GPR" file is present, the displacement scale will automatically be calculated when the file is read. If the user enters new values of a scale, the new scale will remain in effect until a new data file is read. (Changing mining steps will not change the displacement scale if entered by the user.)

Exit MOP Program

Pressing ENTER on this line will exit MOP.

6.2 *Graphics Window*

The MOP graphics window allows the user to obtain a 3-D graphics view of the model geometry. If a ".GPR" file from MAP is present, the user may plot contours of the three principal stresses, plasticity contours, plasticity node indicators and displacement vectors. Screen plots may be rotated and translated. PostScript format hard copies of the screen image an be obtained by pressing the "F10" key. A summary of key functions is given in Table 3.

6.2.1 Model Positioning

While in the graphics window, the user may rotate or translate the model. The magnitude of the rotation or translation can be entered prior to pressing the key to initiate the rotation or translation (as given in Table 3). For rotations, the magnitude is optional, since a default rotation of 10° has been preset. Any entered rotation angle will remain in effect until a new value is entered. There is no preset value for translations, so a magnitude is required. In the case of translation in the plane of the screen, the entered translation will remain in effect until a new value is entered and used. For translations normal to the plane of the screen, a magnitude must be entered each time. Pressing the "F7" key will return the model to the original orientation. The axis icon in the lower left corner indicates orientation only, and is *not* located at the coordinate origin.

6.2.2 Multiple Materials and Regions

In the graphics window, different materials will plot as different colors (unless color by region was selected in the menu). In the case of filled materials, the surface element may plot in one of two colors, depending on the order of the element definition in MIP. The difficulty is that, when modeling fill, void surface elements and fill surface elements occupy the same location in space. To clearly plot either the void or the fill region, the user must hide one material or the other in the menu.

6.2.3 Calculated Data Plots

Pressing "F4" while in the graphics window displays the current plot plane. The user may enter a new plane for plotting, if desired. Plots of output data are only possible on planes parallel to the parallelepiped faces (i,j,k planes). After selecting the desired plane, the user may press "F4" again. The user will then be prompted to press a key corresponding to the plot desired. The following plots are currently available:

S1	plots contours of the maximum principal stress.
S2	plots contours of the intermediate principal stress.
S3	plots contours of the minimum principal stress.
PC	plots areas of plastic failure.
PI	plots nodes where plasticity has occurred.
DISP	plots vectors of nodal displacement.

The program will not plot contours or vectors for nodes which fall within voids. However, if voids exist which are only one nodal point spacing wide, the program is not able to determine the location of the void and will plot contours across these voids. Hiding the foreground in the menu window improves visibility of data plots. The background can also be hidden by turning off the color associated with the object in the background. Then, only the output on the selected plot plane is visible.

6.2.4 Contour Limits

Minimum contour level, maximum contour level and contour interval can be modified in the menu window. In the contour legend, the first and last colors indicate underflow and overflow conditions. Data values outside of the specified range will be plotted in one of these colors. Specified contour intervals are represented by the transition from one color to the next, with the minimum and maximum values occurring at the edge of the underflow and overflow colors.

6.2.5 Displacements

The scale factor for the displacement vectors can also be modified in the menu window. Displacements are initially scaled such that the maximum displacement is at most 70% of the width of the nodal point spacing. Providing a legend with a scale for the displacement vectors is difficult since the view is of the projection of a three-dimensional vector onto the screen. The perspective, therefore, does not always indicate the true length of the displacement, but only the magnitude parallel to the screen. The legend does, however, list the absolute magnitude of the maximum vector of all displacements in the selected i,j,k plane.

6.2.6 Function Keys

F1 The "F1" key will return to the object positioning mode if the user is in the output plane selection node. From the object positioning node, the "F1" key will return to the menu window. It also doubles as a "mouse button 1" when using the cursor to select elements or to select a plot window.

F4 Pressing "F4" selects grid plane for plotting. Pressing "F4" again allows selection of plot type. Pressing "F1" returns to the object position mode.

F5 The "F5" key allows the user to select a specific surface element. After pressing "F5," a cross-hair cursor will appear. The cursor may be moved by using the arrow keys. Pressing "F1" causes the selection of the element under the cursor. The user may then modify the material number of the element, if desired. Note that this is for viewing purposes only, and material number changes are not recorded in the data files.

F7 Pressing "F7" will return the object is in the window to the original centered position and orientation.

F8 Pressing "F8" will activate a cross-hair cursor to be used in modifying the viewing window. The cursor is to be moved by use of the arrow keys to one corner of the new window and "F1" is to be pressed and then moved to the opposite corner and "F1" pressed again. Pressing <esc> will abort the procedure.

F10 Pressing "F10" will cause the generation of a PostScript format file of the screen image.

Table 3

GRAPHICS WINDOW KEY COMMAND SUMMARY

KEY	EFFECT
PgDn	rotate right
Delete	rotate left
Home	rotate up
End	rotate down
PgUp	rotate clockwise
Ins	rotate counterclockwise
up arrow	translate up
down arrow	translate down
left arrow	translate left
right arrow	translate right
ctrl (enter)	translate perpendicular to screen
F1	back-up 1 level
F4	select grid plane for plotting (F4 to plot)
F5	pick element for editing (F1 to pick point)
F7	find image (zoom out)
F8	change plot window (F1 to pick corner)
F10	hard-copy plot

NOTE: All movement keys require a magnitude to be entered first (i.e., 10 PgDn).

7.0 SOFTWARE PROGRAM GUIDE

The program guide contains the complete contents of all the groups of variables in the data structure. The main array A( ) contains parameter groups for surface element variables, surface node variables, volume element variables, volume node variables and connected node variables. All boundary element coefficients are stored in the common block COEFS. Model geometry, property and boundary condition variables are stored in the common block VARS.

MINE3D Model Parameters

Offsets for Surface Element Array — Each element array consists of 11 variables (KELVAR=11). .

Offset	*Parameter*	*Description*
1	KI	i for element
2	KJ	j for element
3	KK	k for element
4	KSET	set number for element
5	KMATER	material number for element
6-9	KNOD	4 node numbers associated with element
10	KEEREG	region number for element
11	KEELIV	element condition code for a mining step: 0 element turned off 1 element turned on

Offsets for Surface Node Array — Each node array consists of 15 variables (KNDVAR=15).

Offset	*Parameter*	*Description*
1	KIN	i for node
2	KJN	j for node
3	KKN	k for node
4	KNCD	normal code indicating presence or absence of surrounding volumes — i.e., parallelepipeds (1 through 254)
5-7	KSOLN	surface solution triple
8-10	KRHS	rhs solution triple
11	KBCCOD	boundary condition code: 1 stress 2 displacement
12	KPLAS	pointer to location in plasticity node list (if KPLAS = 0 then node is elastic only)
13	KENMAT	material number for node
14	KENREG	region number for node
15	KENLIV	node condition code for a mining step: 0 node turned off 1 node turned on

Offsets for Volume Element Array — Each node array consists of 14 variables (KELPVR=14).

Offset	*Parameter*	*Description*
1	KIP	i for volume element
2	KJP	j for volume element
3	KKP	k for volume element
4	KMATP	material number for volume element
5-12	KNODP	8 associated nodes
13	KPEREG	region number for element
14	KPELIV	volume element condition code for a mining step: 0 element turned off 1 element turned on

Offsets for Volume Node Array — Each node array consists of 14 variables (KNDPVR=14).

Offset	*Parameter*	*Description*
1	KINP	i for volume node
2	KJNP	j for volume node
3	KKNP	k for volume node
4-9	KSOLP	6 volume solutions
10	KNCOD	normal code indicating presence or absence of surrounding volumes — i.e., parallelepipeds (0 through 255)
11	KELAS	pointer to location in elastic node list (if KELAS = 0 then node is plastic only)
12	KPNMAT	material number of volume node
13	KPNREG	region number of volume node
14	KPNLIV	volume node condition code for a mining step: 0 node turned off 1 node turned on

Offsets for Node Connection Array — Each connection array consists of 13 variables (KCONN=16).

Offset	*Parameter*	*Description*
0	KPNT	pointer to next pair of connection nodes
1	KN1	number of first node in connection
2	KR1	region number for first node
3	KM1	material number for first node
4	KN2	number of second node in connection
5	KR2	region number for second node
6	KM2	material number for second node
7-12	KINCON	in-situ stresses for second node (stresses for first node are in ffsig array)
13-15	KDOCON	incremental displacements at connection nodes, used for displacement boundary condition

Logical Unit Numbers

LUN	*Parameter*	*Description*
1	MINPUT	input file
5	MINKEY	keyboard
6	MOUT	screen output
7	MOFILE	output file
11	LUNGO	graphics output file
22	MSRF	direct access file for STRVOL coefficients
23	MDSRF	direct access file for DISSRF coefficients
24	MSVOL	direct access file for STRVOL coefficients
25	MDVOL	direct access file for DISVOL coefficients

Array Limits

Parameter	*Present Value*	*Description*
IJKMAX	*	maximum number of nodes must be > IMAX*JMAX*KMAX
NMATMX	10	maximum number of materials
ISIZE	15*IJKMAX	size of A array
NELMAX	8*IJKMAX	maximum number of surface or volume elemens allowed
IVNMAX	**	maximum number of plastic volume nodes (stores plastic correction)

* IJKMAX = 300 for 16 Mb RAM — Estimate maximum size by:

memory required to run MAP ≈ 0.04 + 0.024 * IJKMAX [Mbytes]

** IVNMAX = 300 for 16 Mb RAM

Coefficient Common Block (COEFS) Variables

STRSRF(3,6,4)	surface stress coefficients
INSRF(3, NELMAX)	record numbers of surface stress coefficients
IRSRF	current last record of STRSRF file
DISSRF(3,3,4)	surface displacement coefficients
INDSRF(3,NELMAX)	record numbers of surface displacement coefficients
IRDSRF	current last record of DISSRF file
STRSLF(3,254)	surface stress coefficients self-effects for 254 node combinations
DISSLF(3,254)	surface displacement coefficients self-effects for 254 node combinations
STRVOL(6,6,8)	volume node stress coefficients
INSVOL(NELMAX)	record numbers of volume stress coefficients
IRSVOL	current last record of STRVOL file
DISVOL(6,3,8)	volume node displacement coefficients
INDVOL(NELMAX)	record number of volume displacement coefficients
IRDVOL	current last record of DISVOL file
STRSLFV(6,255)	volume stress coefficients self-effects for 255 node combinations
DISSLFV(3,255)	volume displacement coefficients self-effects for 255 node combinations
COESLF(3,6,254)	constant term to coefficient of fictitious force component
ISLF(254)	normal code number check: = 0 if not treated previously = 1 if treated previously

Plastic Stress Correction Common Block (CORR) Variable

STRCOR(6,IVNMAX)	stores plastic stress correction

Model Geometry Common Block (VARS) Variables

LSTEP	load step number
MSTEP	mining step number
FRAC	fraction of load per step
IOS	error code (0 = no error)
NMAT	number of materials in model
NREG	number of regions in model
NLD	number of load steps for plastic solution (not used)
NIT	number of iterations for solution
RPMAX	relaxation parameter max. (not used)
RPMIN	relaxation parameter min.
TOL	tolerance on iterative solution
PROP(16,NMATMX)	material properties: 1 Young's modulus 2 Poisson's ratio 3 derived property for kernal 4 derived property for kernal 5 derived property for kernal 6 derived property for kernal 7 cohesion 8 friction angle 9 dilation angle 10 tensile strength 11 derived plasticity property 12 derived plasticity property 13 derived plasticity property 14 derived plasticity property 15 derived plasticity property 16 displacement scaling factor
FSTRESS(12,NMATMX)	in-situ stress variables: sxx = fstress(1,*) + fstress(2,*) * z sxy = fstress(3,*) + fstress(4,*) * z sxz = fstress(5,*) + fstress(6,*) * z syy = fstress(7,*) + fstress(8,*) * z syz = fstress(9,*) + fstress(10,*) * z szz = fstress(11,*) + fstress(12,*) * z
XYZN(3,3,254)	254 different possible node coordinate systems
XYZ(3,3,3)	3 different element coordinate systems
XCOORD(3,4)	4 points defining main parallelepiped: 1 at i=j=k=0 2 at i=imax, j=k=0 3 at j=jmax, i=k=0 4 at k=kmax, i=j=0

IMAX	maximum number of elements in i direction
JMAX	maximum number of elements in j direction
KMAX	maximum number of elements in k direction
NEL	total number of surface elements
NNOD	total number of surface nodes
A(ISIZE)	main array for element and node storage
NELBEG	pointer to surface elements in A array
NODBEG	pointer to surface nodes in A array
SIG(6,IJKMAX)	stress components for all nodes SXX = SIG(1,*) SXY = SIG(2,*) SXZ = SIG(3,*) SYY = SIG(4,*) SYZ = SIG(5,*) SZZ = SIG(6,*)
FFSIG(6,IJKMAX)	in-situ stress components at all nodes
DISP(3,IJKMAX)	displacement components for all nodes XD = DISP(1,*) YD = DISP(2,*) ZD = DISP(3,*)
RIVEC(3)	i direction vector
RJVEC(3)	j direction vector
RKVEC(3)	k direction vector
NODPBG	pointer to volume nodes in A array
NELPBG	pointer to volume elements in A array
NELP	total number of volume elements
NNODP	total number of volume nodes
IADCON	pointer to connected nodes list in A array
NODREG(IJKMAX)	region numbers for all nodes
NODMAT(IJKMAX)	material numbers for all nodes
NITE	number of iterations over elastic nodes before iterations for plastic nodes within iteration NIT
NITP	= 0 if elastic analysis = 1 if plastic analysis
EPSOF	corner offset magnitude
EPFAC	corner offset factor

MINE3D Text Common Block (BEMTXT) Variables

LNAME	problem title
LDATE	date
LTIME	time
FNAME	file name

8.0 EXAMPLE PROBLEMS

8.1 *Square-Shaped Drift*

A 6 m x 6 m square shaped drift 24 m in length is modeled as a simple example to demonstrate the application of MINE3D. Both elastic and plastic simulations are performed.

The first step is to create the model in MIP. Upon execution of MIP, the screen display as shown in Fig. 13 should appear. The user has the option to specify a new grid, restore an existing workspace, ask for help, or quit.

For this problem, a new grid is created, so the user selects the first menu item and moves to the menu shown in Fig. 14. First, the user defines the overall grid size — in this case, 12 m in the x-direction, 12 m in the y-direction, and 24 m in the z-direction. The origin of the grid is at x=y=z=0.0, and the grid axes are orthogonal, which are the default settings, so these menu items do not have to be called. The number of elements in the grid are selected to be 8 elements in the i-, j- and k-directions. Note that MIP requires that the number of ***elements*** be specified, ***not*** nodes. The total number of parallelepiped nodes for this model is then 9 x 9 x 9 = 729 nodes.

Once the grid is defined, the user may view the model by displaying 2-D slices along any of the grid planes in the i-, j- or k-directions. The user excavates the drift by moving the cursor to each of the middle 4 x 4 = 16 elements in slice 1 on the x′-y′ plane and changing the elements to "mined-out" (black) by pressing the left-hand mouse button (see Fig. 15). In order to extend the excavation through the grid, the "mined-out" pattern on slice 1 is copied to the other 7 slices by moving the cursor to the ***copy pattern*** menu item and pressing the mouse button for the remaining 7 slices.

The excavation can be displayed in other views. For example, by pressing the ***show y′-z′ plane*** and the ***Jump to*** menu items, the user can move to slice 4 in the y′-z′ plane. The excavation is shown in Fig. 16.

A grid printout is selected for the k-4 plane by jumping to slice 4 of the x′-y′ plane and turning on ***print request***.

Next, properties, far-field stresses and solution parameters are defined through the ***props, data, etc.*** menu item. Field stresses are defined for region number 1 (the infinite body), and the following values are specified:

xx-stress:	9.0	[MPa]
yy-stress:	9.0	[MPa]
zz-stress:	9.0	[MPa]

Material properties are then defined for material number 7 (the infinite body material). The properties specified are:

Young's modulus: 1000.0 [MPa]
Poisson's ratio: 0.3

For the elastic analysis, the following solution parameters are selected:

maximum number of iterations:	1
maximum number of elastic passes per iteration:	100
plastic analysis? (0=no; 1=yes):	0
relaxation parameter — maximum value:	0.80
corner offset fraction:	0.01
solution tolerance:	0.001
energy calculation (0=no; 1=yes)	1
sign convention? (0=com. pos.; 1=stress.pos.)	0
mining step number to start calculation:	1

Now, returning to the main menu, the ***file functions*** item is selected. First, the workspace is saved under the file name "TEST1" (the extension .MIP is automatically added). Then the MAP data file is written and given the name "TEST1E.DAT". The user can now return to the main menu and quit MIP.

File "TEST1E.DAT" should be loaded into MOP to view the model geometry. Figure 17 presents a copy of a PostScript$^{(R)}$ plot generated with MOP.

The computational analysis is now performed by reading "TEST1E.DAT" into MAP. This model contains 160 surface elements and 162 surface nodes. A total of 16 elastic passes are required for the solution error to fall below the prescribed tolerance. Two files are created: "TEST1E.OUT" and "TEST1E.GPR". The first file contains a summary of the conditions specified for this analysis plus a printout of the results along plane K-4.

The second file contains results for all the parallelepiped nodes in the grid. This file can be read into MOP. (Note that the .DAT file must also be available and that the file is read into MOP without the extension.) Example displacement and stress contour plots are shown in Figs. 18 and 19. In these plots, the foreground is hidden from view and a wireframe mode view is selected.

A check of the energy calculations for the square tunnel can also be made. The strain energy density can be calculated from the pre-mining stresses:

$$U_m = \phi(\sigma) = \frac{\left[\sigma_1^2 + \sigma_2^2 + \sigma_3^2 - 2\nu\,(\sigma_1\sigma_2 + \sigma_2\sigma_3 + \sigma_3\sigma_1)\right]}{2E} \qquad (9)$$

$$= 0.15\ \frac{MJ}{m^3}$$

This agrees with the MINE3D calculation for U_m divided by the excavation volume. The energy component W_b is determined by summing the primitive tractions on the surface to be excavated times the induced displacements at the corresponding nodes:

$$W_b = \frac{1}{2} \sum_{i=1}^{n} \left(T_i^{(P)} \, u_i^{(i)} \, A_i \right) \tag{10}$$

where n =number of surface nodes,

$T_i^{(P)}$ = traction components from primitive stresses at each node,
= $\sigma_{ij} \, n_j$, where σ_{ij} = initial stress.

$u_i^{(i)}$ = induced displacements for each node
= $u_i - u_i^{(P)}$, and

A_i = surface area of each node.

Positive values for W_b indicate a decrease in stored strain energy in the body; therefore, energy is leaving the host material. For the example problem, W_b = 0.300 MJ.

A plasticity analysis can be made by making a new data file based upon the "TEST1.MIP" workspace file. After reading "TEST1.MIP" into MIP, the user must now add plastic volume elements around the opening. First, it is necessary to go to the ***select pick color*** item in the main menu. The screen shown in Fig. 20 will appear. The "P" box is selected with the cursor and the user is returned to the main menu. Two rows of elements are turned plastic with the cursor. Figure 21 shows the opening and plastic elements for slice 1. This slice is copied to all other slices, as before. Figure 22 shows the view in the y'-z' plane.

In the ***props, data, etc.*** menu, the user must define plasticity material properties. In this case, cohesion = 5 MPa and Young's modulus = 1000 MPa, and Poisson's ratio = 0.3, as before. All other properties all left at the default values of zero.

Also, in the ***solution parameters*** menu, the following parameters are changed:

maximum number of iterations:	100
plasticity analysis? (0=no; 1=yes):	1
energy calculation? (0=no; 1=yes):	0

A new MAP data file is created from the ***file functions*** menu, "TEST1P.DAT". This file can now be read into MAP for analysis. In addition to 160 surface elements and 162 surface nodes, there are now 384 volume elements and 648 volume nodes. The total memory requirement to load MAP is still the same as before, since the total number of parallelepiped nodes is still 729. The solution this time requires 46 plastic iterations and a total of 294 elastic passes. (The number of iterations may vary slightly depending on differences in machine tolerances.)

The graphical results (from "TEST1P.GPR") are presented in Figs. 23 through 25. In addition to stress contour and displacement vector plots, a plastic node plot is also shown. This plot indicates the extent of plasticity which develops in the analysis.

Fig. 13 MIP Initial Screen Display

Fig. 14 Grid Dimensions Menu

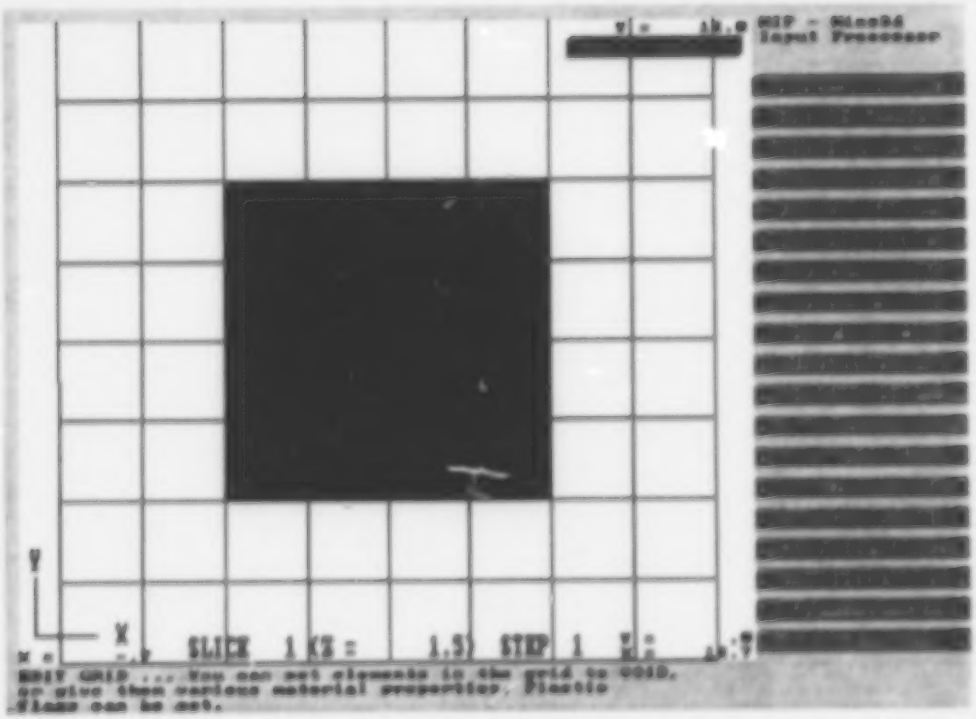


Fig. 15 Excavation in Slice 1 of x'-y' Plane of Grid

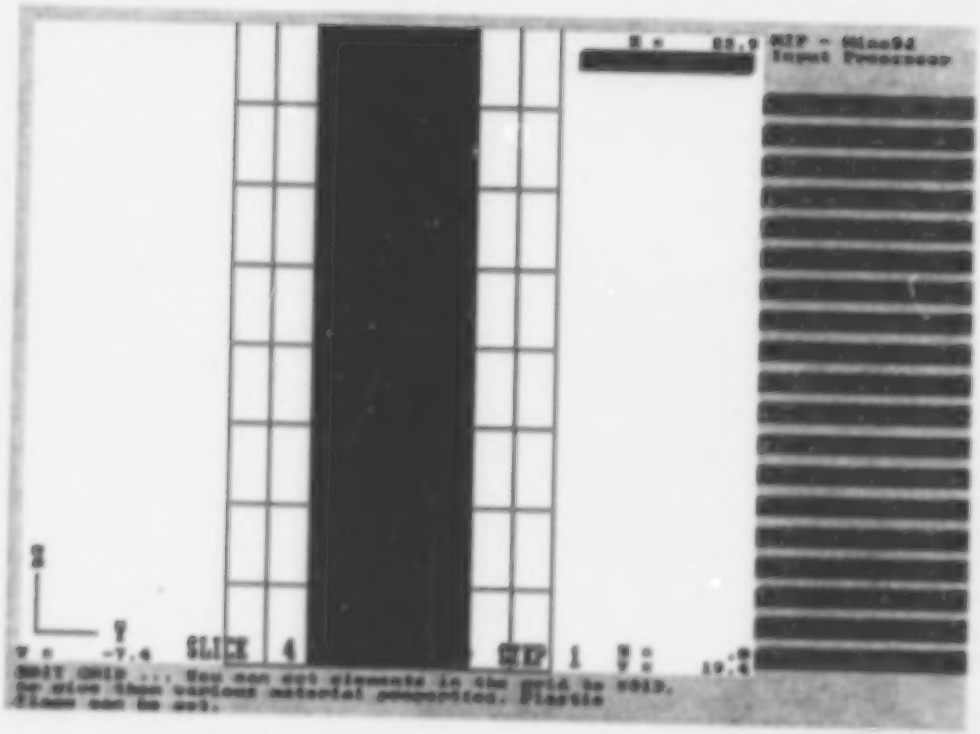


Fig. 16 Excavation in Slice 4 of y'-z' Plane of Grid

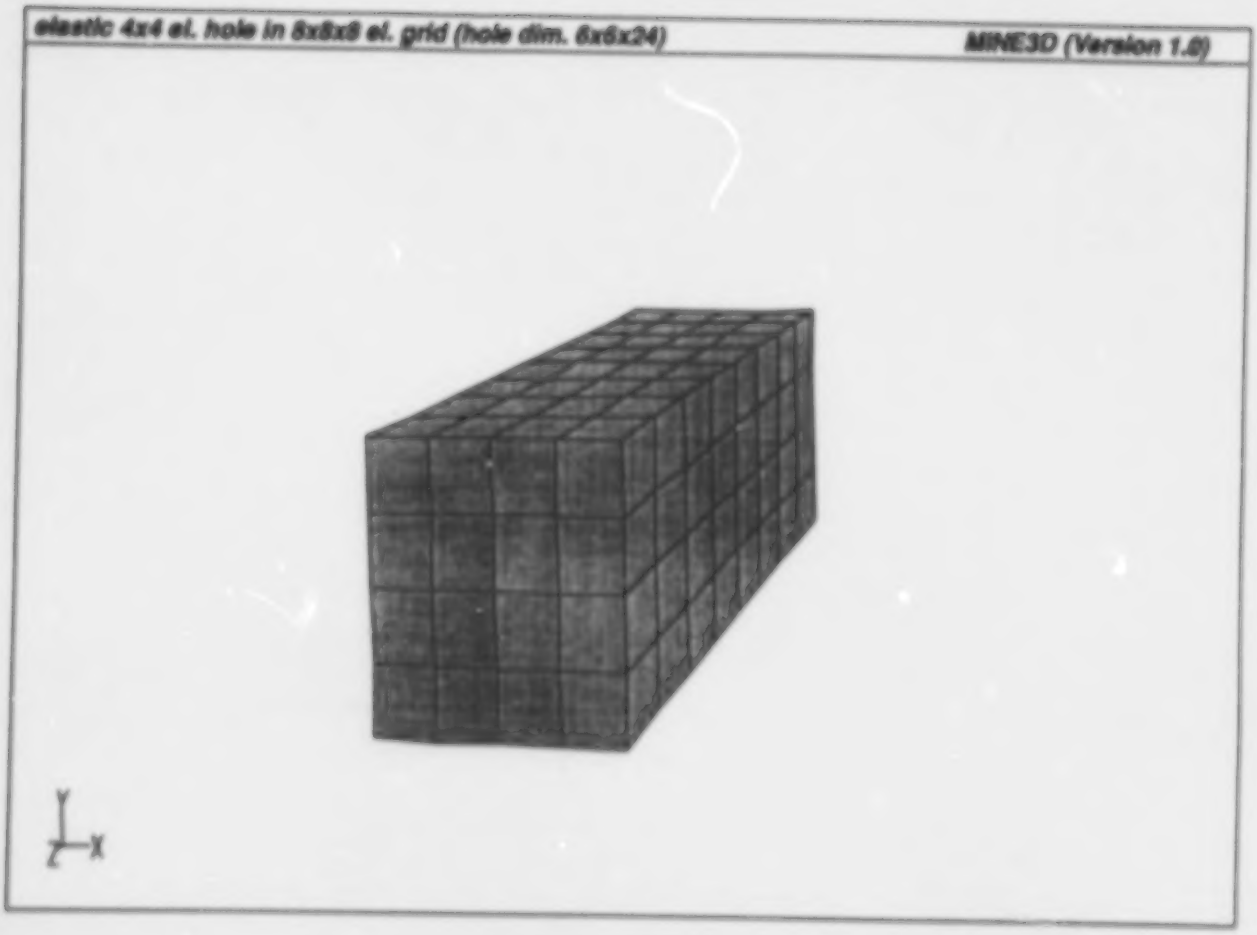


Fig. 17 3-D Geometry Plot from MOP

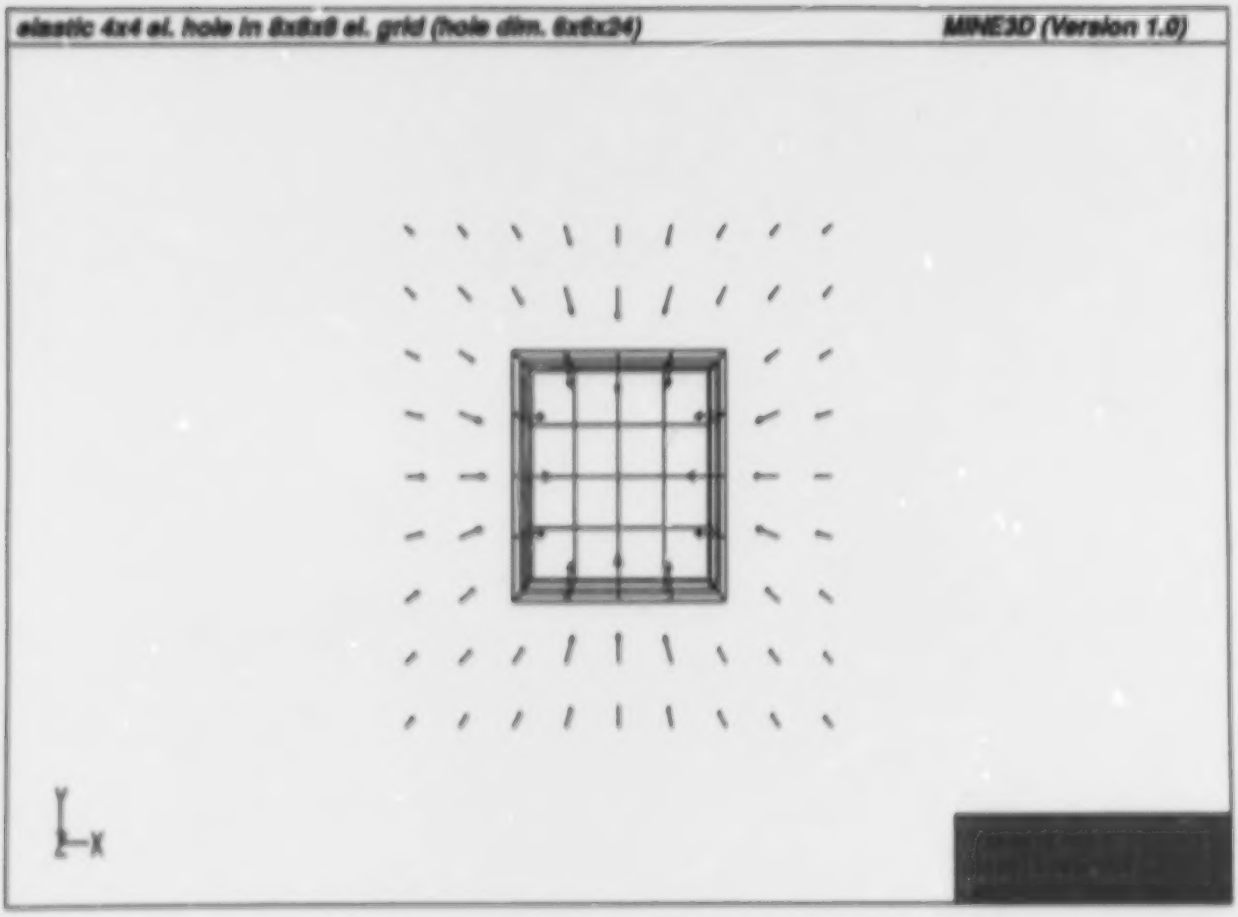


Fig. 18 Displacement Vectors on K-5 Plane for Elastic Analysis

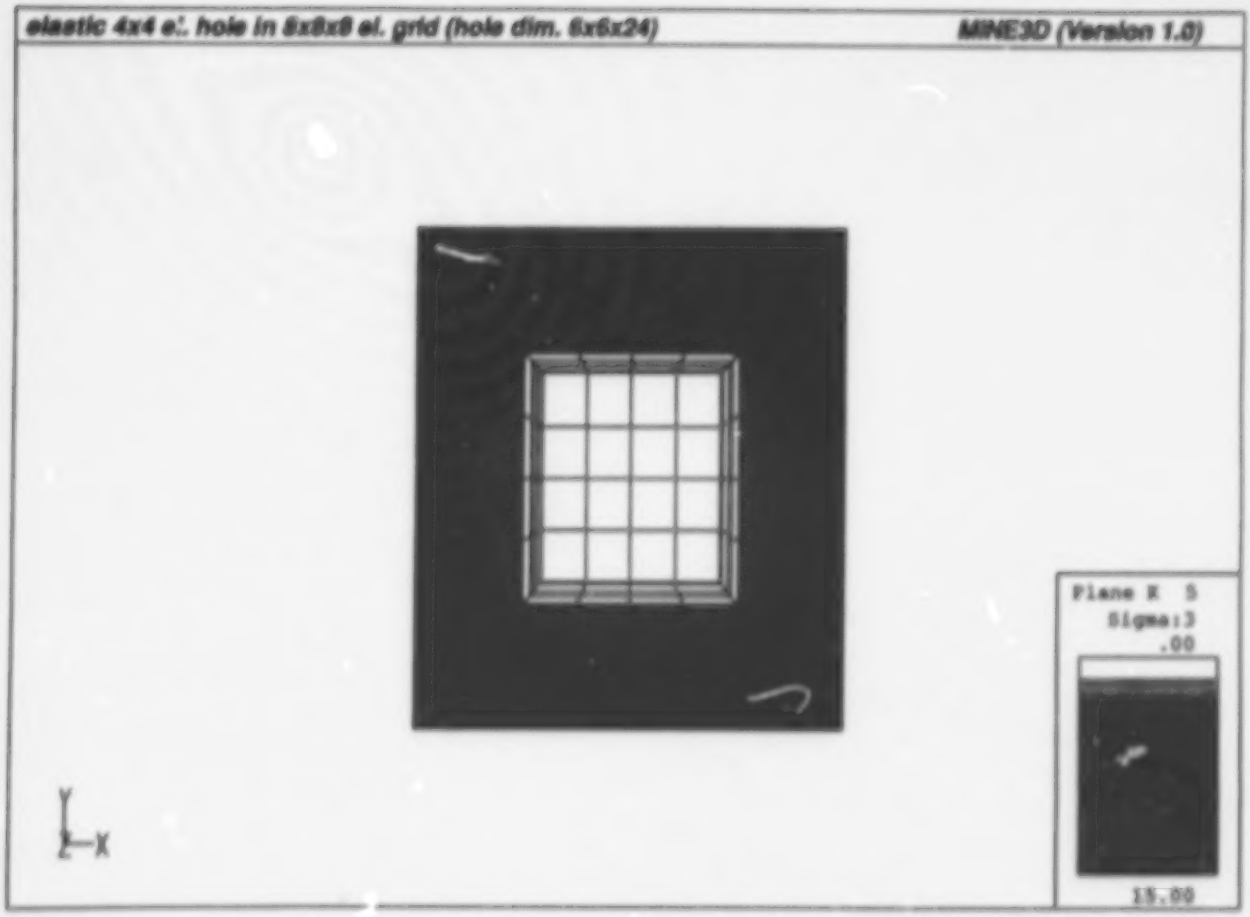


Fig. 19 σ_3 *Stress Contours on K-5 Plane for Elastic Analysis*

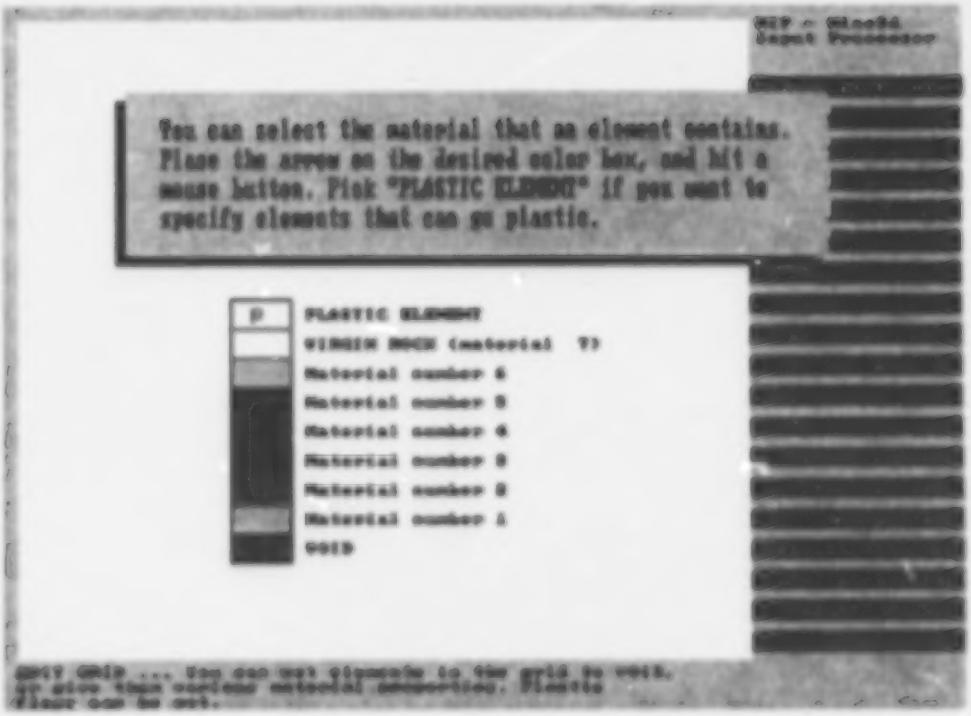

You can select the material that an element contains.
Place the arrow on the desired color box, and hit a
mouse button. Pick "PLASTIC ELEMENT" if you want to
specify elements that can go plastic.
P
PLASTIC ELEMENT
VIRGIN ROCK (material 7)
Material number 6
Material number 5
Material number 4
Material number 3
Material number 2
Material number 1
VOID

Fig. 20 Select Element Type Menu

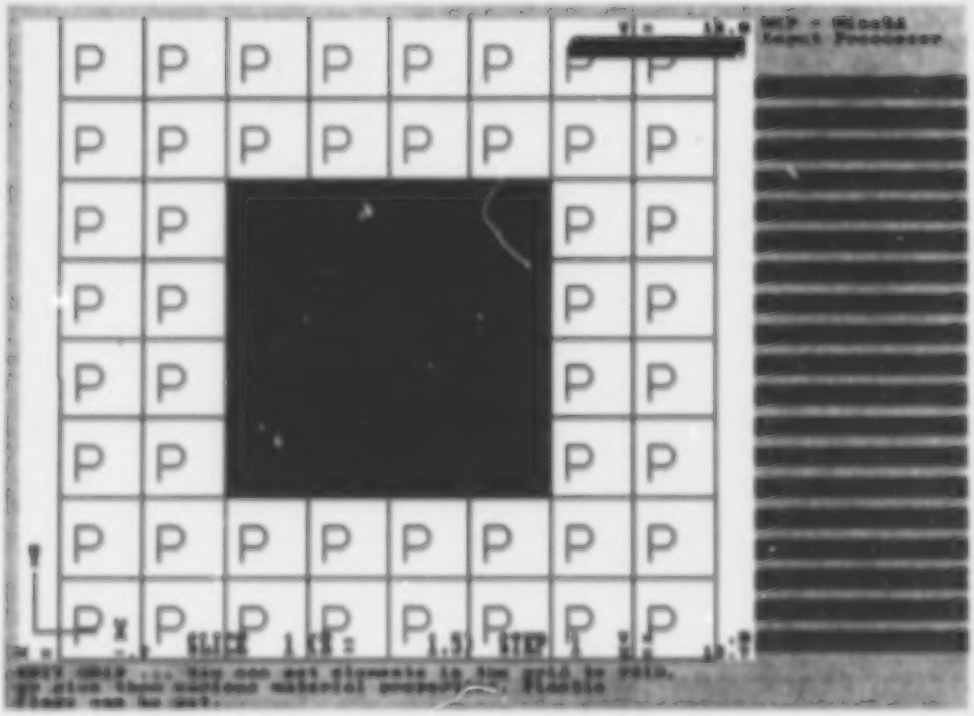


Fig. 21 Excavation and Plastic Elements in Slice 1 of x'-y' Plane of Grid

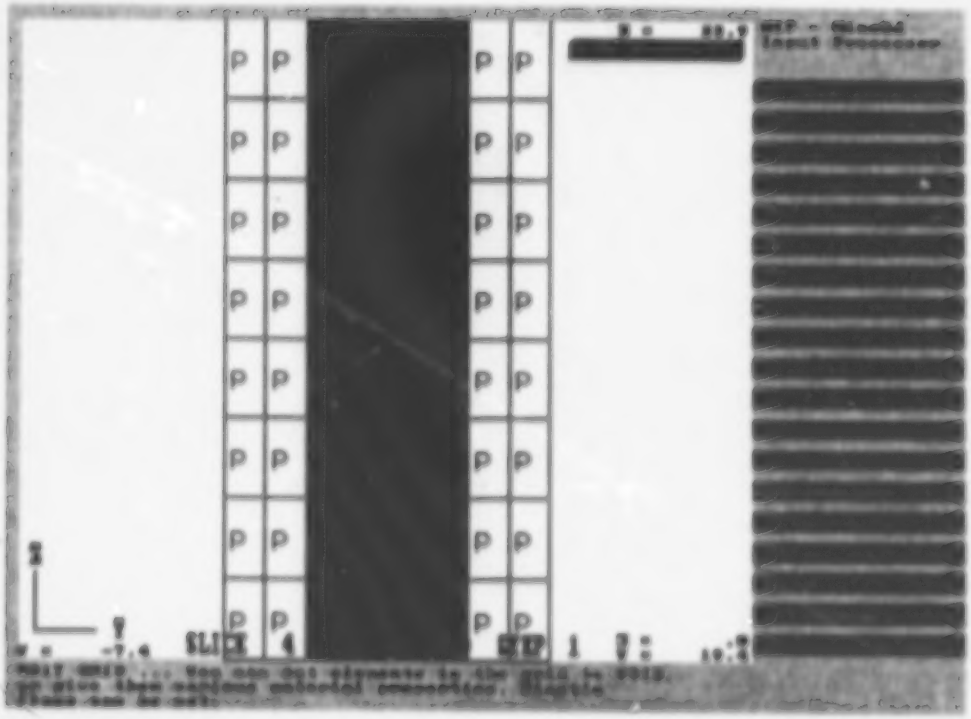


Fig. 22 Excavation and Plastic Elements in Slice 4 of y'-z' Plane of Grid

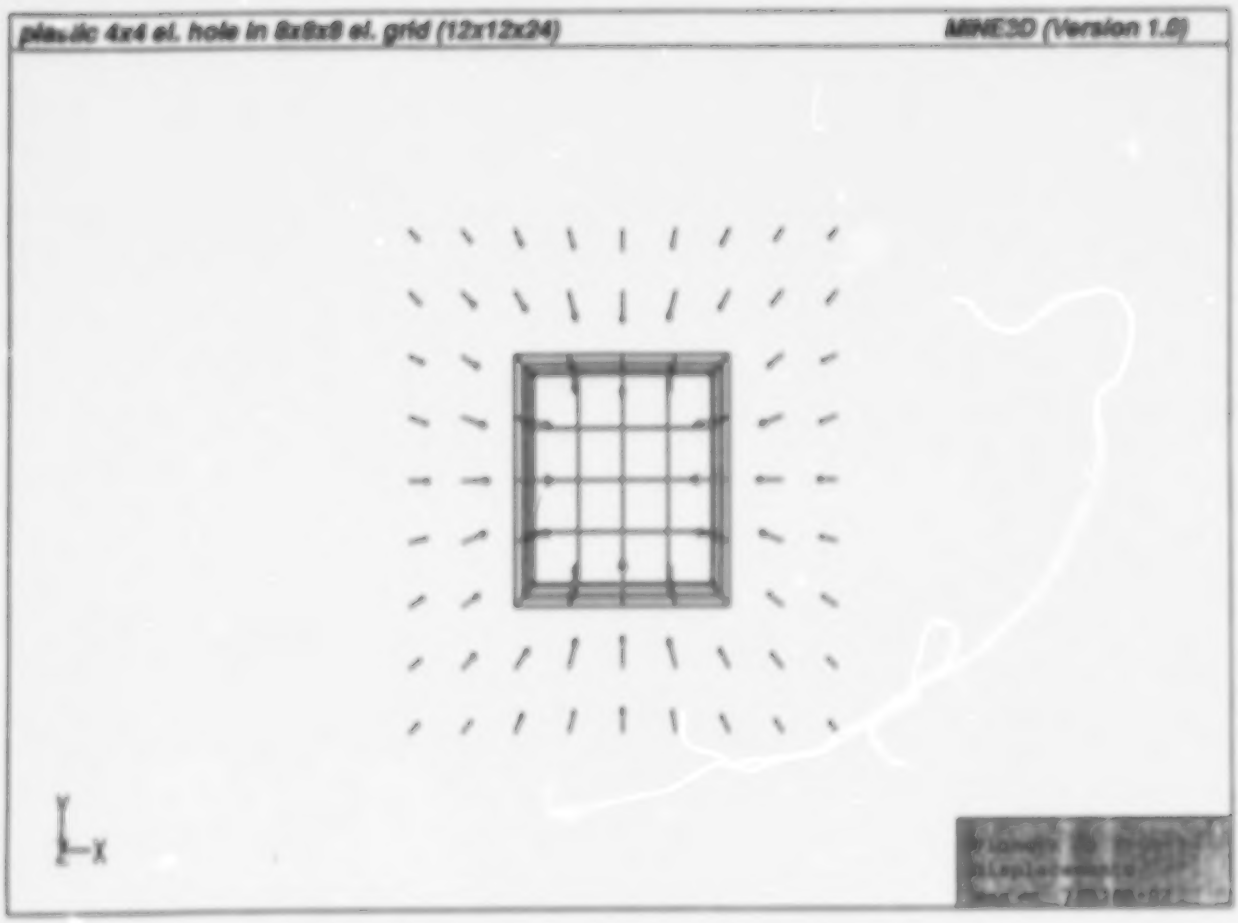


Fig. 23 Displacement Vectors on X-S Plane for Plastic Analysis

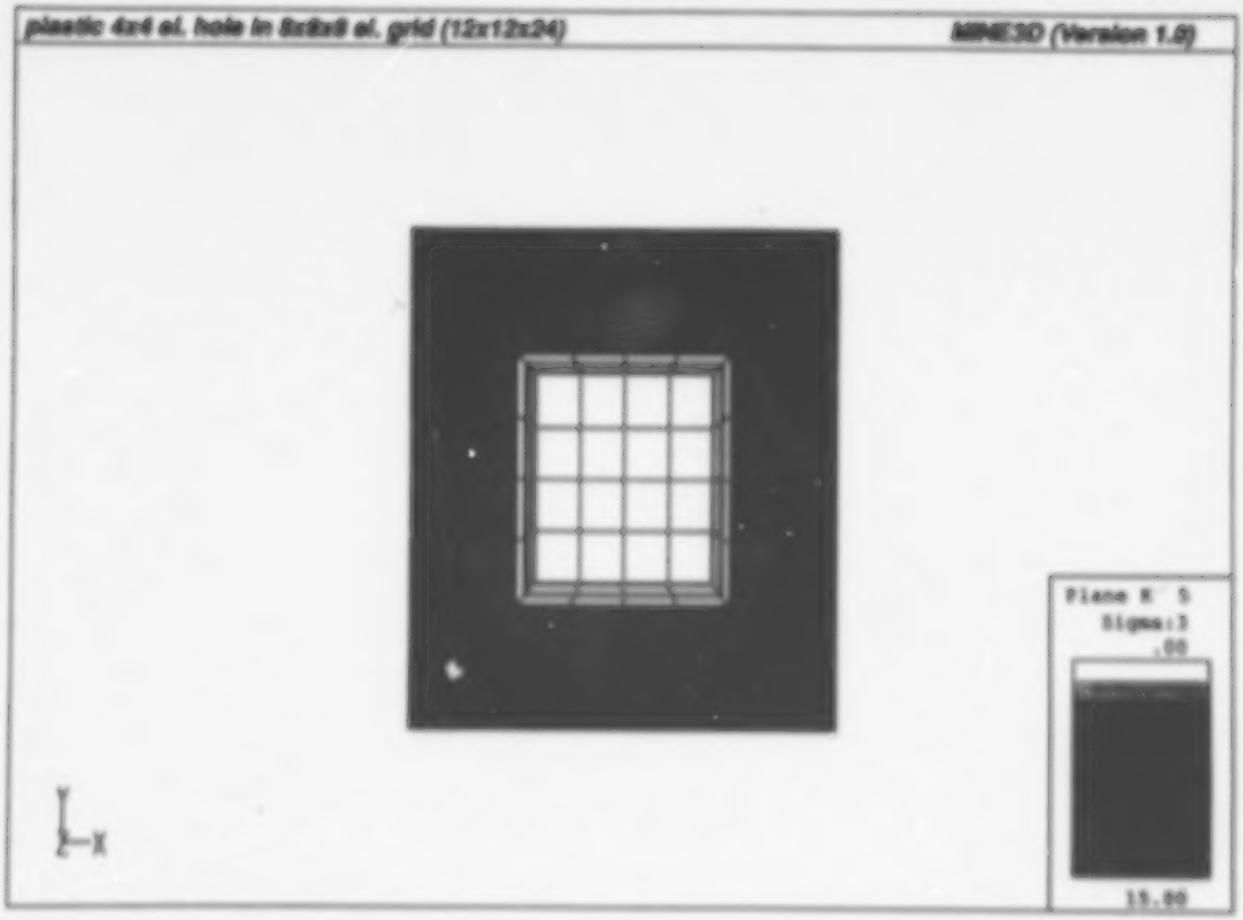


Fig. 34 σ_3 Stress Contours on K-5 Plane for Plastic Analysis

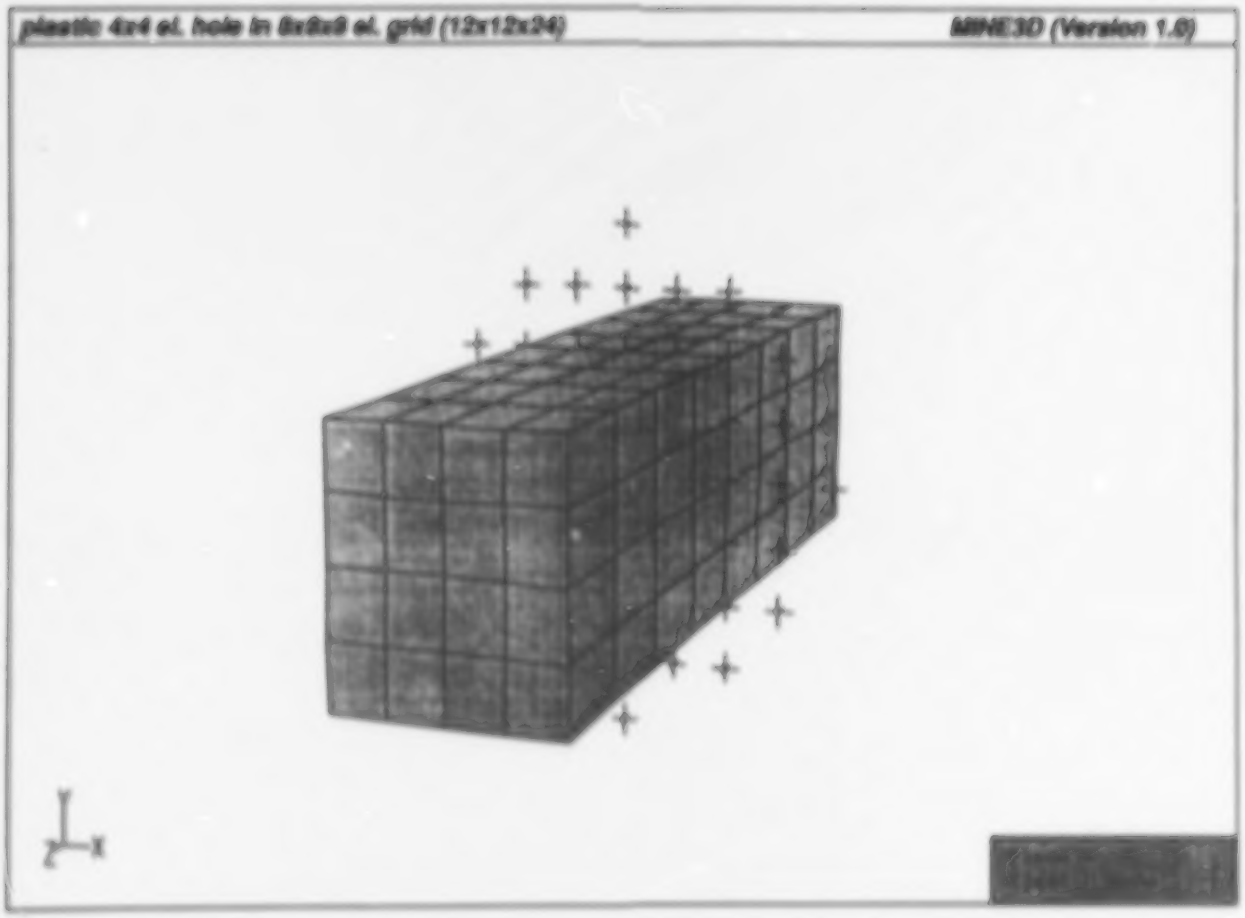


Fig. 25 Plastic Nodes on K-5 Plane for Plastic Analysis

8.2 *Excavation / Fill Stopes*

This example demonstrated the creations of a MINE3D model based upon AutoCAD drawings of a mining pattern. The pattern consists of two stopes which are mined sequentially. In Step 1, the first stope is excavated; in Step 2, this stope contains fill material and the second stope is excavated.

Before the AutoCAD file can be accessed, the parallelepiped grid must be defined. The overall grid size is selected to encompass the area to be analyzed. In this example, the grid size is 150 ft in the x-direction, 130 ft in the 7-direction, and 80 ft in the z-direction. The origin of the grid must be specified to position the grid within the AutoCAD drawing. In this example, the origin is

x-value of grid origin	3965.0 ft
y-value of grid origin	6350.0 ft
z value of grid origin	6232.0 ft

The number of elements selected for this model grid are:

number of elements in local x-direction 20

number of elements in local y-direction 20

number of elements in local z-direction 6

This grid is designed to cover a limited region at a specified level (6700 level) of the mine.

Now the user enters the main menu and selects the *file functions* menu. From here, an AutoCAD "DXF" file is read into MIP. Once this file is entered, the user selects the *show AutoCAD layers* menu item in the main menu and can now select which drawings to display. Figure 26 shows the listing of layers for the given file. The 6700 layer is turned on. Figure 27 shows the drawing of the 6700 level and the parallelepiped grid overlaid.

At this stage, the user may reposition the grid or change the number of elements or grid size by selecting the *re-specify grid* menu item. Once the user is satisfied, the process continues in much the same manner as before for the square-shaped drift example. Now, though, two mining steps are defined. In the first step, the user mines out the access way and the first stope (see Fig. 28). The pattern is specified on slices 2 through 5 in the x'-y' plane of the grid, for a stope height of 53.3 ft.

The user now selects *next mining step* and mines out the second stope on slice 2. The *select pick color* item is then used to select material number 1. The first stope is now "filled" with material 1. The result is as shown in Fig. 29. Slice 2 at mining step 2 is now repeated for Slices 3, 4 and 5 by using the *copy pattern* function.

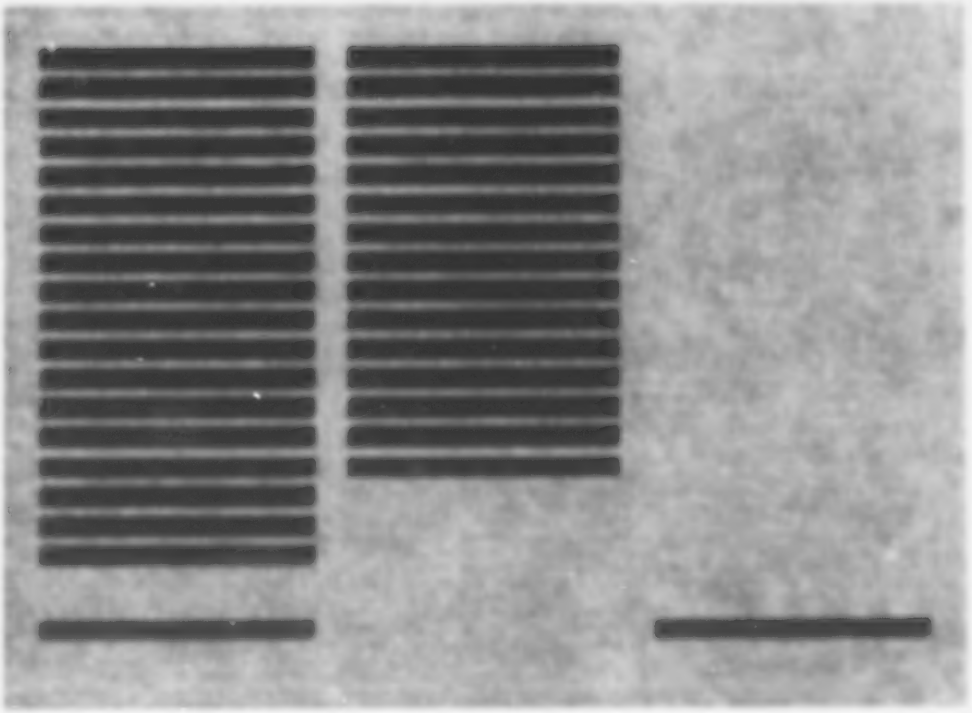


Fig. 26 Layers Within an AutoCAD "DXF" File

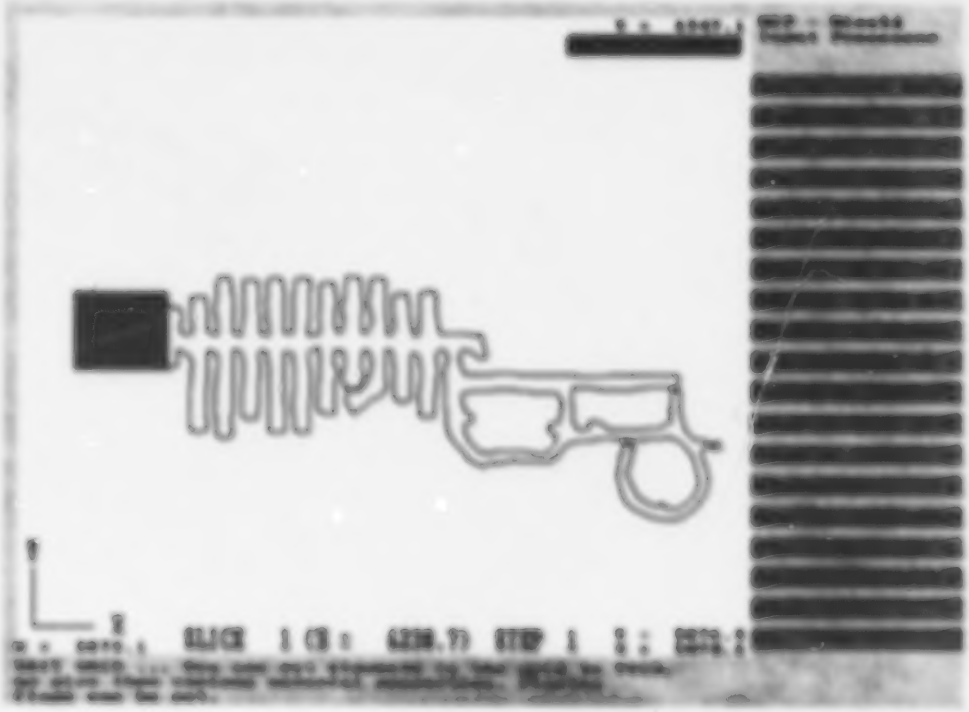


Fig. 27(a) MINE3D Grid Overlays AutoCAD Drawing: View of Full Drawing

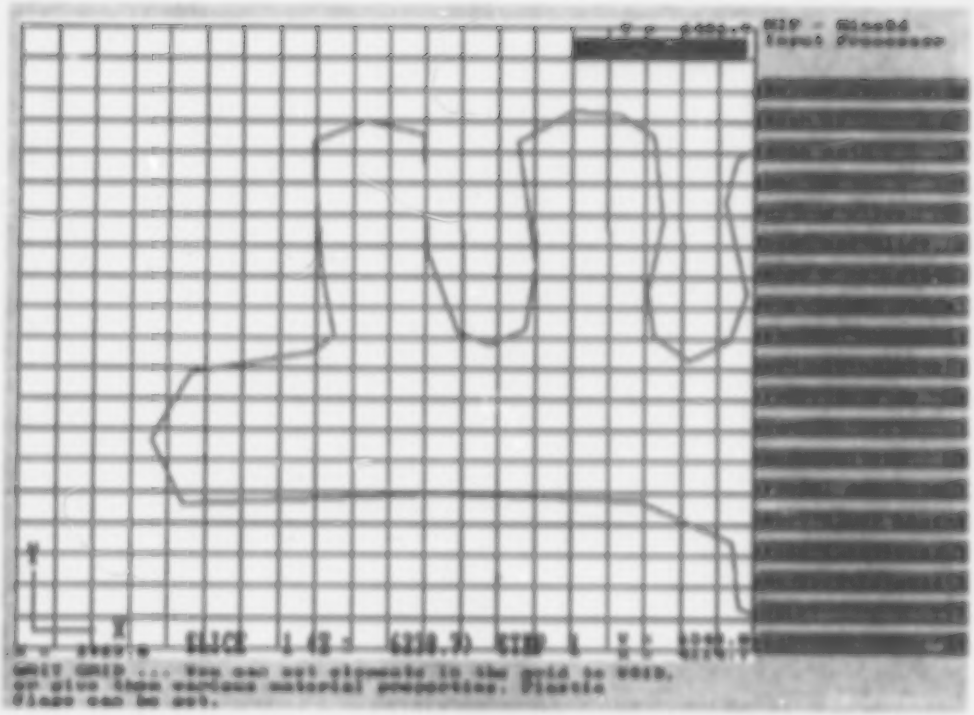


Fig. 27(b) MINE3D Grid Overlays AutoCAD Drawing: Close-Up (Zoom-In) View

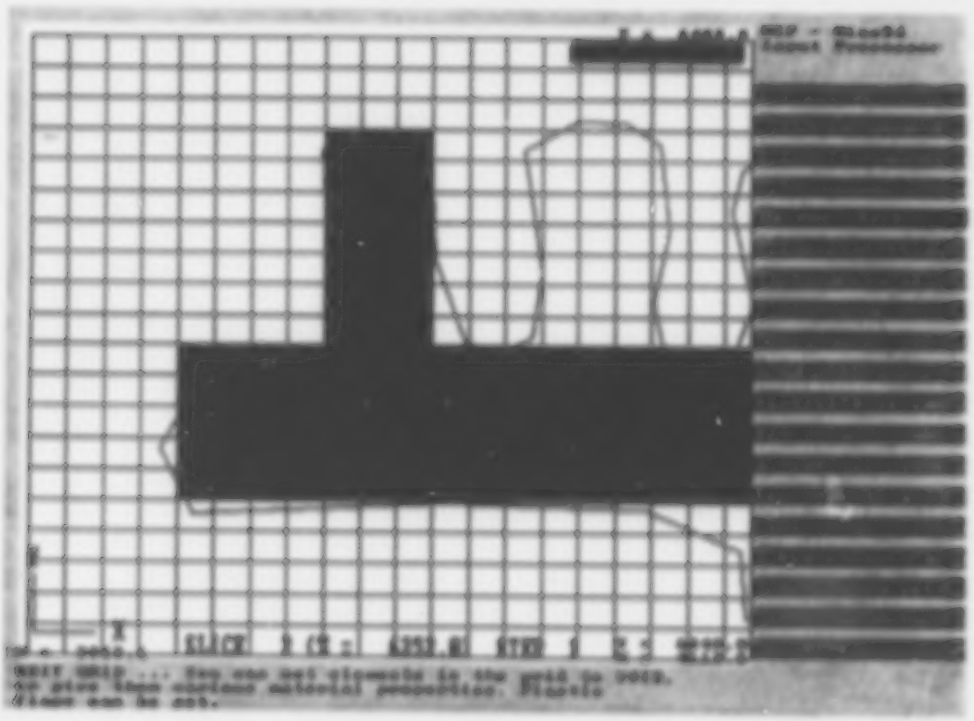


Fig. 28 Mining Step 1

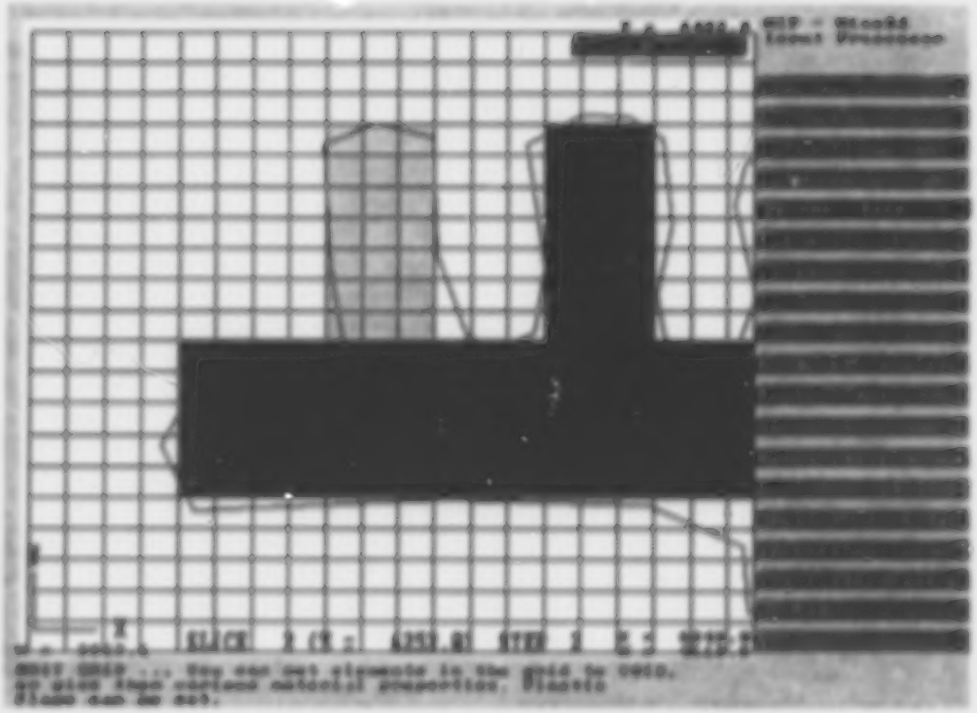


Fig. 29 Mining Step 2

Properties, stresses and solution parameters are now entered through the ***props, data etc.*** function. The field stresses are defined for region number 1:

xx-stress:	1.95	[x10^6 psf]
yy-stress:	1.25	[x10^6 psf]
zz-stress:	1.25	[x10^6 psf]

Region number 2 is assigned to the fill region in the first stope. For this example, the initial stresses are assumed equal to zero in region 2.

The material properties are defined for material number 7 (the infinite body material):

Young's Modulus:	1420	[x10^6 psf]
Poisson's Ratio:	0.3	

and for material number 1 (the fill material):

Young's Modulus:	100	[x10^6 psf]
Poisson's Ratio:	0.3	

Remember that the Poisson's ratios for all materials *must* be equal for the multiple material logic to work.

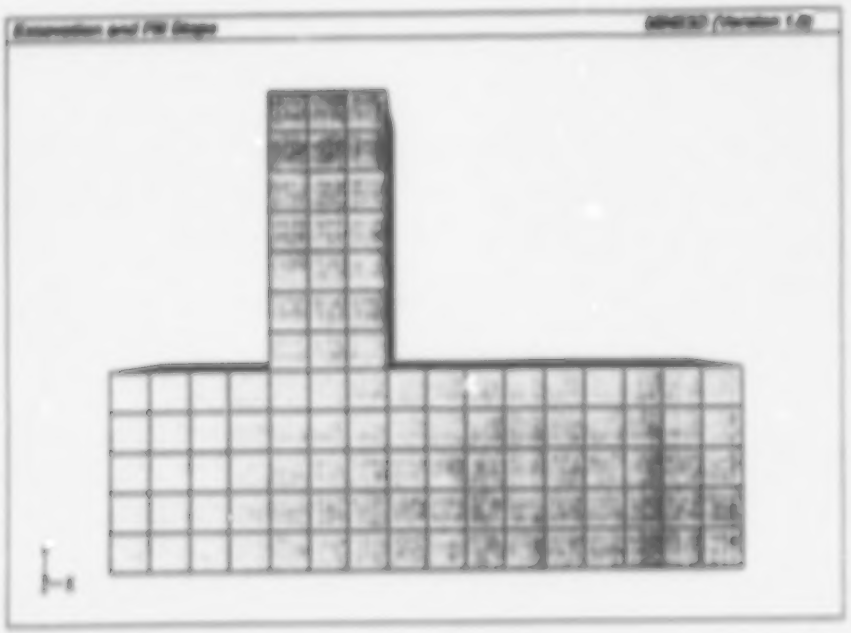
All default solution parameters are used for this elastic analysis.

The workspace file is saved with the filename "L67(.MIP)" and the MAP data file with the filename "L67.DAT".

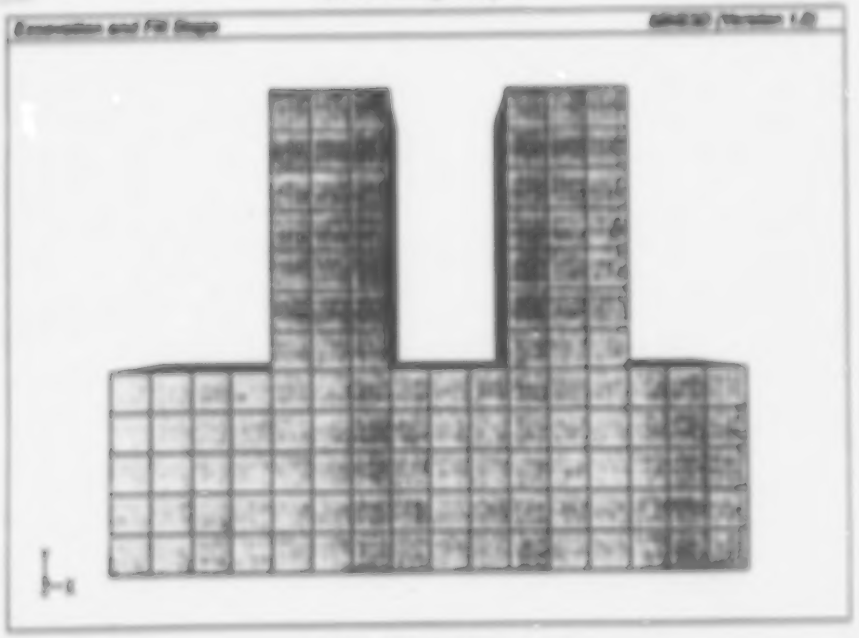
The data file is then read into MOP. The 3-D perspective views for mining steps 1 and 2 are shown in Figs. 30(a) and (b).

The MAP analysis is now performed. This model contains a total of 3087 parallelepiped nodes, 822 surface elements and 814 surface nodes. During the first mining step, 428 surface nodes are active. During the second step, 650 surface nodes are active and 242 surface nodes are connected. Step 1 requires 14 elastic passes to reach the solution tolerance and step 2 requires 18 passes.

Displacement vector plots and stress contour plot for stope 1 are shown in Figs. 31 through 33. Displacement vectors are plotted along the K-4 and J-13 planes through the grid, and the σ_3 stress contour plot is made along the K-4 plane. Similar plots are given in Figs. 34 through 36 for mining step 2.



(a) Mining Step 1



(b) Mining Step 2

Fig. 30 3-D Geometry Plot of Excavation and Fill Stopes

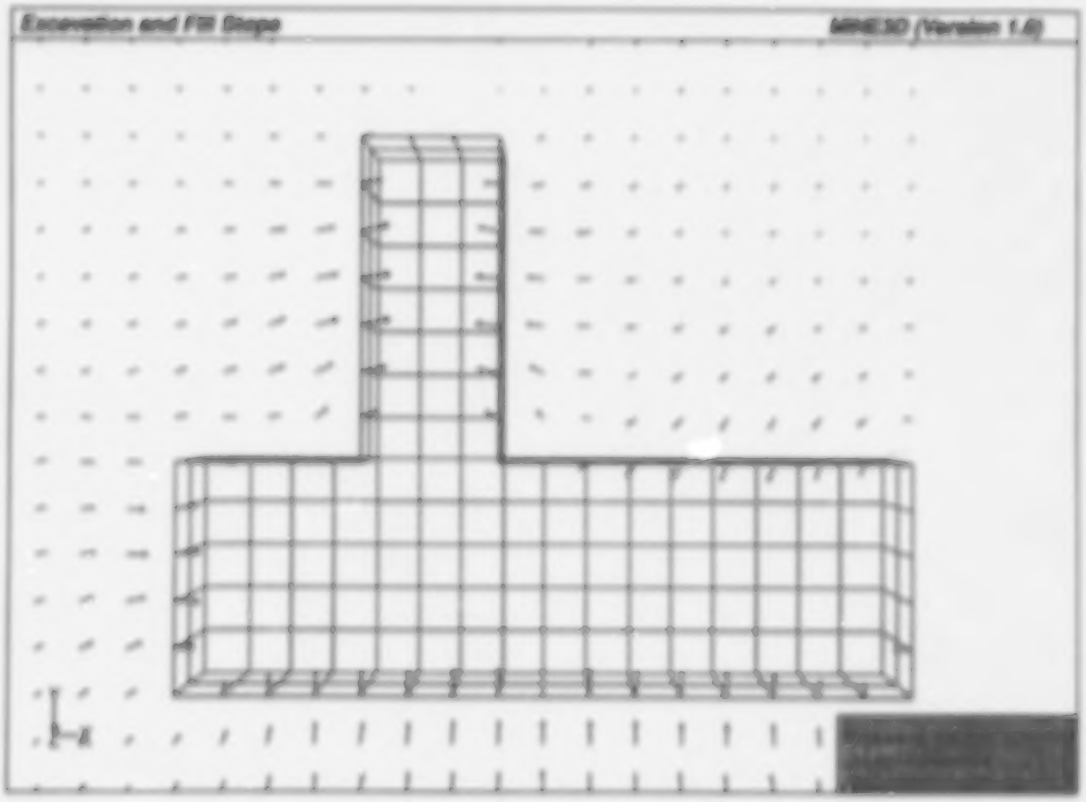


Fig. 31 Displacement Vectors on K-4 Plane for Mining Step 1

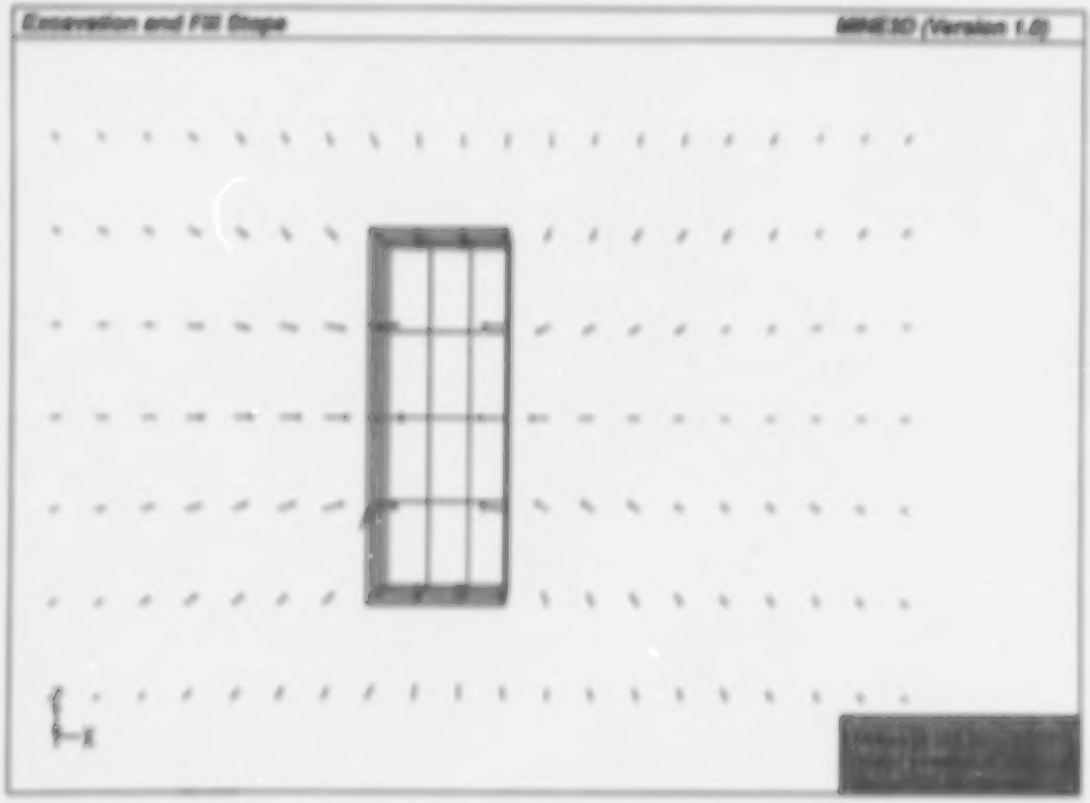


Fig. 32 Displacement Vectors on J-13 Plane for Mining Step 1

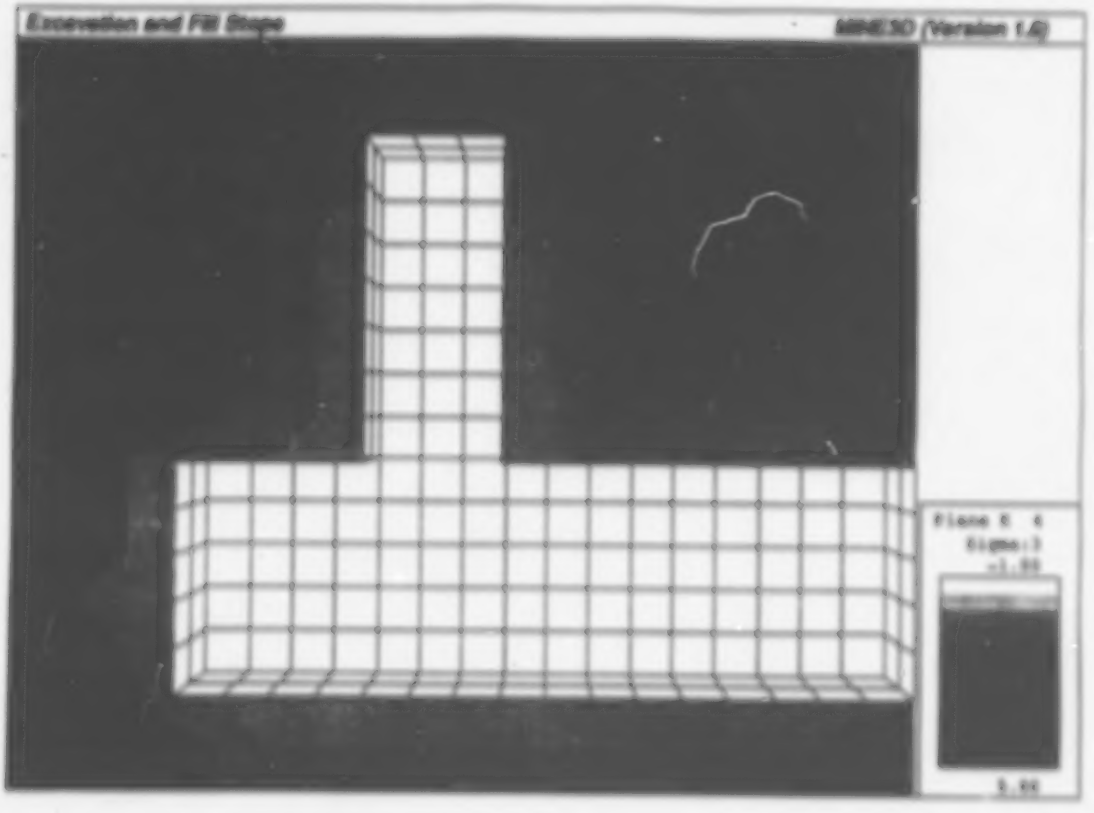


Fig. 33 σ_3 Stress Contours on K-4 Plane For Mining Step 1

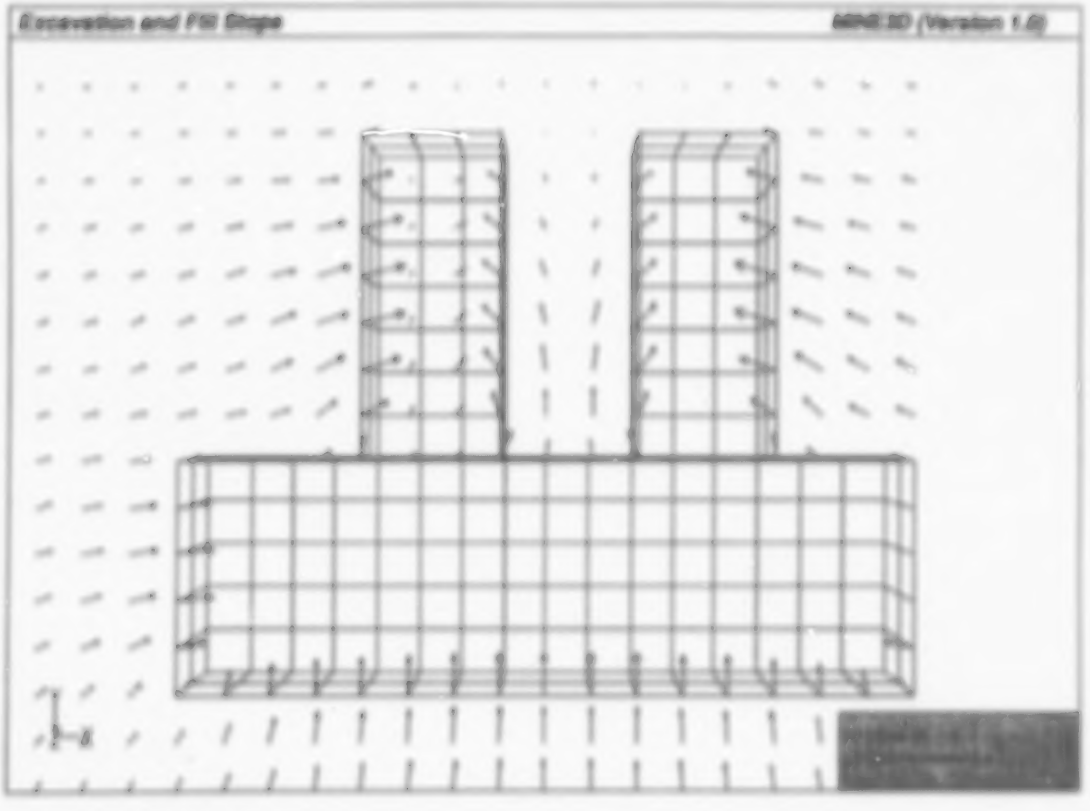


Fig. 34 Displacement Vectors on K-4 Plane For Mining Step 2

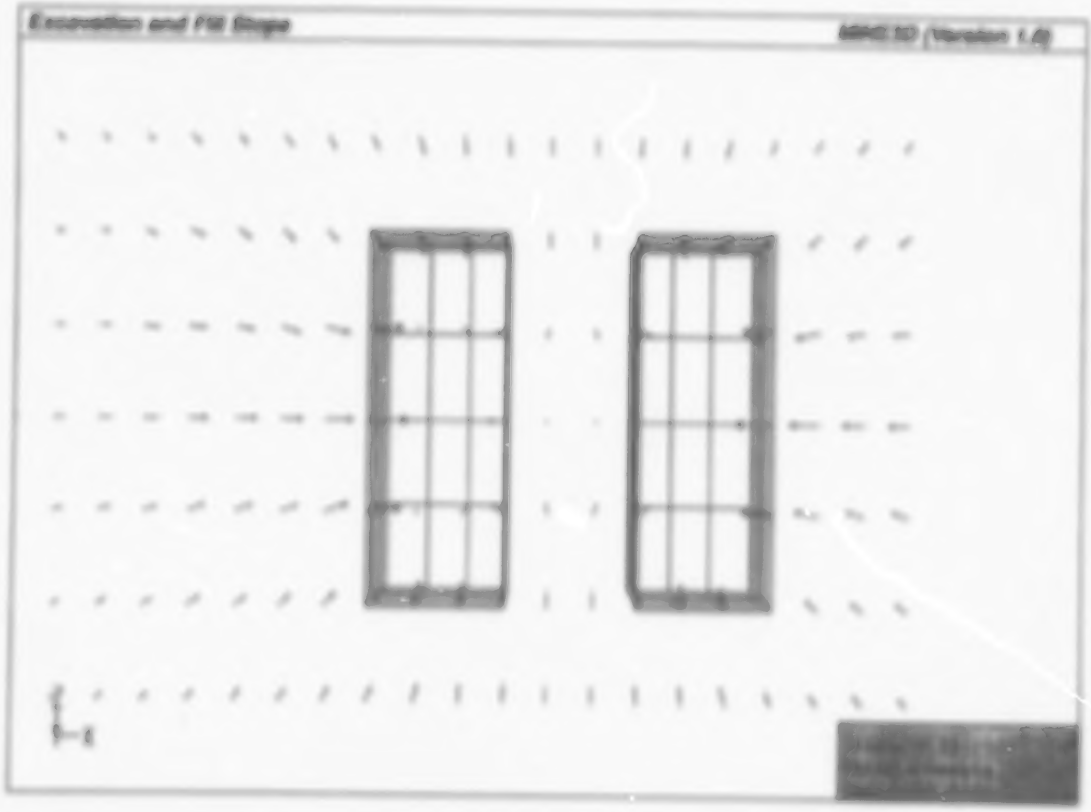


Fig. 35 Displacement Vectors on J-13 Plane For Mining Step 2

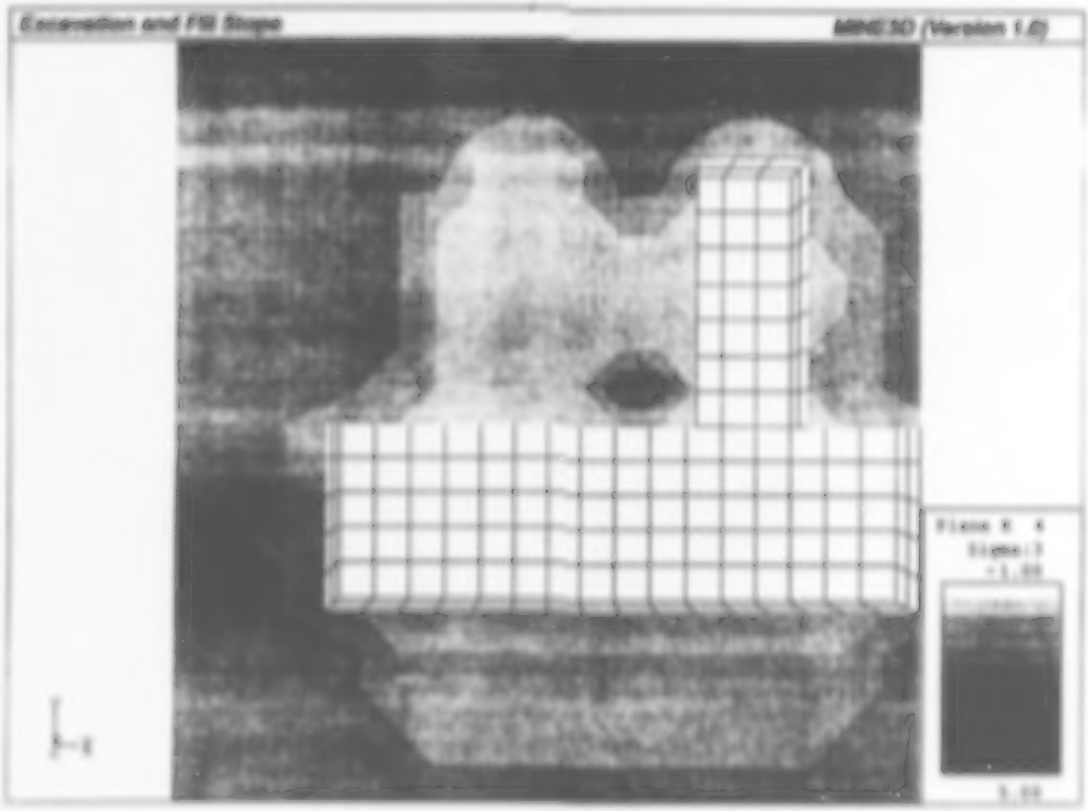


Fig. 36 σ_3 Stress Contours on K-d Plane For Mining Step 3

9.0 REFERENCES

Banerjee, P. K. and R. Butterfield. (1981) ***Boundary Element Methods in Engineering Science***. London: McGraw-Hill Book Company (UK) Limited.

Dahlquist, G., and A. Bjorck. (1974) ***Numerical Methods***. Englewood Cliffs, New Jersey: Prentice-Hall.

Salamon, M.D.G. (1984) "Energy Considerations in Rock Mechanics: Fundamental Results," *J. S. Afr. Inst. Min. Metall.*, *84*(8), 233-246.

APPENDIX I

MINE3D SOURCE FILES

The following are supplied as source files for program MINE3D. The program is divided into three separate codes: MIP (MINE3D Input Processor), MAP (MINE3D Analysis Program), and MOP (MINE3D Output Processor). The files for each code are listed separately. The files with extension .FOR are written in FORTRAN 77 and those with extension .ASM are written in assembly language for an 80386 microcomputer.

The FORTRAN and assembler files for MIP and MOP are written for operation on an 80386-based microcomputer running DOS 4.x, using the SVS FORTRAN 386 compiler with PHARLAP linker, and DOS extender. The screen graphics support for MIP and MOP is handled through the graphics library files in "IG.FOR" and "U386.ASM" written by P.A. Cundall (PAC Plotting Package, by Itasca Consulting Group, Inc.).

The FORTRAN files for MAP are written for operation on either an 80386-based microcomputer or on a workstation such as a SUNSPARCStation. MAP can be easily transferred to any computer operating in FORTRAN 77.

MIP

File	Description
BUTIL.FOR	utility routines specific to MIP
IGMIP.FOR	MIP plotting routines
IO.FOR	files-handling routines
MENU.FOR	graphics menu routines
MIP.FOR	main program routines
UTIL.FOR	non-specific utility routines
ASYS386.ASM	access DOS system routines
MEM.ASM	automatic memory-sizing for 80386
M386.ASM	assembly routines specific to MIP
MIPCOM.INC MATCOM.INC	common block files

MAP

File	Description
ASSEMBLE.FOR	routines to assemble coefficients
ASYS386.FOR	access Sun keyboard no wait routine (replace with ASYS386.ASM for 80386)
DISCAL.FOR	routines to calculate displacement at a node
ENERGY.FOR	energy release routines
INPUT.FOR	input routines
KERNEL.FOR	kernel functions for disp and stress
MAIN.FOR	main program routines
MSIN1.FOR	routines for mining step changes
MUTIL.FOR	MINE3D utility routines
OUTPUT.FOR	output routines
PREP.FOR	generates element, coord. and property information
QUAD.FOR	routines for Gauss quadrature
RECORD.FOR	controls direct access files
SAVE.FOR	saves all coefficients (not in use)
SELF.FOR	contains self effect routines
SOLVE.FOR	SOR solution routines
STRSCAL.FOR	routines to calculate stress at a node
SUTIL.FOR	access Sun systems routines
UTIL.FOR	non-specific utility routines
VOLPLAS.FOR	plasticity solution routines
VUTIL.FOR	vector and tensor transformations, etc.
BEMCOM.INC	main common block file

MOP

File	Description
ARROW.FOR	displacement vector arrow routine
GRFIGC.FOR	basic graphics routines
INIT.FOR	initialization routines
KEYBD.FOR	keyboard function routines
MOP.FOR	main program routines
PS.FOR	hardcopy plotting routines
VLIB.FOR	utility routines specific to MOP
UTIL.FOR	non-specific utility routines
VO.FOR	view utility routines
VT.FOR	MOP plotting routines
ASYS386.ASM	access DOS keyboard no wait routine
COMMON.BEM COMMON.2D COMMON.3D COMMON.IO COMMON.R0D COMMON.R1D COMMON.R2D COMMON.R3D COMMON.VIE COMMON.VR	common block files
MCC.INC	
PSCOM.INC	

PAC GRAPHICS LIBRARY FILES (for MIP and MOP)

File	Description
IG.FOR	general plotting routines
U386.ASM	graphics utilities routines for 8038 in protected mode
IGPCOM.INC	common block files

APPENDIX II

COLOR PLOTS FROM PostScript$^{(R)}$ FILES